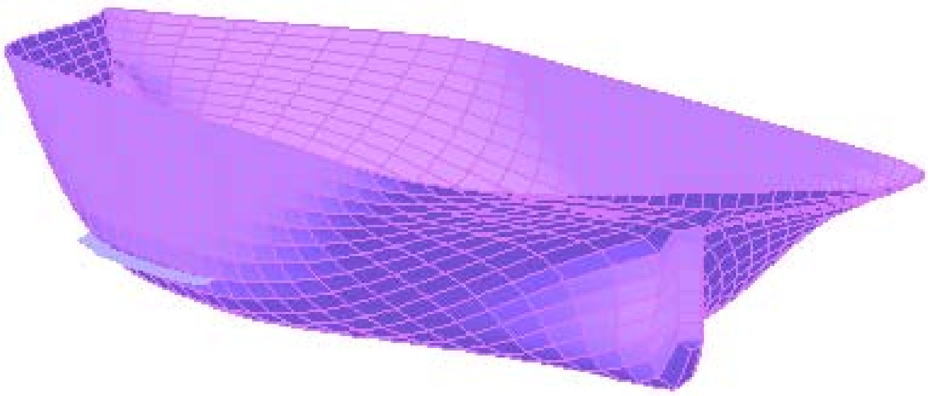
SESAM USER MANUAL

HYDROD



Wave load & stability analysis
of
fixed and floating structures

Det Norske Veritas

SESAM
User Manual
HydroD

Wave load & stability analysis of
fixed and floating structures

8 April 2011

Valid from program version 4.5

Developed and Marketed by
DET NORSKE VERITAS

Published by:

Det Norske Veritas Software
Veritasveien 1
N-1322 HØVIK
Norway

Telephone: +47 67 57 99 00

Facsimile: +47 67 57 72 72

E-mail, sales: software.sesam@dnv.com

E-mail, support: software.support@dnv.com

Website: www.dnvsoftware.com

HydroD User Manual
Version 4.5

Table of Contents

1 INTRODUCTION ... 5

1.1 HydroD Overview ... 5
1.2 HydroD Analysis Types ... 6
1.3 HydroD in the Sesam System ... 6
1.4 How to read this Manual ... 6
1.5 Terminology and Notation ... 6
1.6 Release Notes and Status List ... 6
1.7 HydroD Extensions ... 7

2 APPLICATION TOOLS ... 8

2.1 Feature / license selection ... 8
2.2 Application Window ... 9
2.3 Data Organization ... 10
2.4 Context Sensitive Actions ... 12
2.5 Wizards ... 12
2.6 Graphical Interaction ... 17
2.7 Selection ... 17
2.7.1 Add Clients to Selection ... 18
2.7.2 Apply Property to Selection ... 19
2.8 Tool tips ... 19
2.9 Reporting ... 20
2.10 Copy/Paste ... 22
2.11 Draw style Settings ... 22
2.12 View Options ... 24
2.13 View Toolbars ... 26
2.13.1 The Main Toolbar ... 26
2.13.2 The View Manipulations ... 26
2.13.3 The Label Toolbar ... 26
2.13.4 The Selection Toolbar ... 26
2.13.5 The HydroD Tools Toolbar ... 26
2.13.6 The Views Toolbar ... 27
2.13.7 The Hydro Models Toolbar ... 27
2.13.8 The Analysis Toolbar ... 28
2.13.9 The Dimension Toolbar ... 28
2.13.10 The Snap Toolbar ... 28
2.13.11 The Environment Toolbar ... 29
2.14 View Settings ... 29
2.15 Colour Coding of Properties ... 30
2.16 Units ... 31
2.17 Scripting ... 32
2.18 Save Clean JS ... 34
2.19 Batch ... 34
2.20 Prewad JNL Files ... 34
2.21 Graph Controls ... 35
2.22 Grid Controls ... 36
2.23 Dialog New/Edit Functionality ... 37
2.24 Late Definition Functionality ... 37

3 FEATURES OF HYDROD 38

3.1 Hydrostatic Analysis 38
3.1.1 Hydrostatic Data 38
3.1.2 GZ curve 38
3.1.3 Righting Moment 38
3.1.4 Wind Heeling Moment 39
3.1.5 Openings 39
3.1.6 Cross Section Data 39
3.1.7 Hydrostatic Balancing of Compartments 39
3.1.8 Hydrostatic Rule Checks 40
3.2 Hydrodynamic Analysis 40
3.2.1 Hydrostatic Data 40
3.2.2 Inertia Properties 40
3.2.3 Global Response 41
3.2.4 Load Transfer 41
3.2.5 Other Results 42
3.3 3D Visualization 42
3.3.1 Display Control 42
3.3.2 Dimension 43
3.4 Interactive Modelling 43
3.5 Transformation of Models 44
3.6 Data Checking 45

4 USER'S GUIDE TO HYDROD 46

4.1 The Modelling Loop 46
4.2 Coordinate Systems 46
4.3 Finite Element Models 47
4.3.1 Panel FEM 47
4.3.2 Morison FEM 48
4.3.3 Mass FEM 48
4.3.4 Structure FEM 48
4.3.5 Second Order Free Surface FEM 49
4.3.6 Offbody Points FEM 49
4.3.7 Legal Elements Types 49
4.4 Model Configurations 50
4.4.1 Panel Model Configuration 50
4.4.2 Morison Model Configuration 51
4.4.3 Composite Model Configuration 51
4.4.4 Dual Model Configuration 52
4.4.5 Multi-body Configuration 52
4.5 Output Files 53
4.5.1 Hydrodynamic Results Interface File 53
4.5.2 Loads Interface File 53
4.5.3 Analysis Control Data File for Structural Analysis 53
4.5.4 Wadam Print File 53
4.5.5 Wasim Print Files 54
4.6 Load Case Numbering 54
4.7 Program Limitations 55

5 DIALOG DESCRIPTION 56

5.1 Workspace 56
5.1.1 New Workspace 56
5.1.2 Old Workspace 56
5.2 Environment Modelling 57
5.2.1 Data Organization 57
5.2.2 Water Data Types 57
5.2.2.1 Jonswap Wave Spectrum 57
5.2.2.2 Bretschneider (two parameter Pierson-Moskowitz) Wave Spectrums 58

5.2.2.3 Torsethaugen Wave Spectrums 58
5.2.2.4 Spreading Functions 58
5.2.2.5 Discretized Spreading Functions 59
5.2.2.6 Current Profiles 59
5.2.2.7 Frequency Sets 59
5.2.2.8 Phase Sets 60
5.2.2.9 Wave Height Functions 60
5.2.2.10 Wave Height Surfaces 61
5.2.2.11 Regular wave sets 61
5.2.3 Directions Data Types 62
5.2.3.1 Directions 62
5.2.3.2 Direction Sets 62
5.2.4 Location Data Types 62
5.2.4.1 Locations 62
5.2.4.2 Frequency Domain Conditions 62
5.2.4.3 Time Domain Conditions 63
5.2.4.4 Sea States 63
5.2.4.5 Irregular Time Conditions 63
5.2.5 Air Data Types 64
5.2.5.1 Exponential Wind Profile 64
5.2.5.2 IMO MODU Wind Profile 64
5.3 Hydro Models 65
5.3.1 Data Organization 65
5.3.2 Compartments Properties 66
5.3.2.1 Deck Tanks 66
5.3.2.2 Filling Fractions 67
5.3.2.3 Flooded Tanks 67
5.3.2.4 Fluids 67
5.3.2.5 Permeabilities 68
5.3.3 Morison Properties 68
5.3.4 Rudder Properties 68
5.3.5 Thruster Forces 68
5.3.6 Wind Properties 68
5.3.6.1 Drag Block Coefficient Curve 69
5.3.6.2 Drag Coefficient Curve 69
5.3.7 Section Model 70
5.3.8 Import DXF geometry to section model 70
5.3.9 PLN file format 71
5.3.10 Panel Model 72
5.3.10.1 Pressure Panels 72
5.3.10.2 Strip Model 73
5.3.10.3 Bilge Keel 74
5.3.11 Morison Model 76
5.3.11.1 Anchor Elements 76
5.3.11.2 Anchor Sections 76
5.3.11.3 Pressure Area Elements 77
5.3.11.4 Pressure Areas 77
5.3.11.5 TLP Elements 78
5.3.11.6 TLP Sections 78
5.3.11.7 Morison 3D Elements 78
5.3.11.8 Morison 3D Sections 79
5.3.11.9 Morison (2D) Sections 79
5.3.11.10 Element Correspondence 80
5.3.12 Load Cross Sections 82
5.3.12.1 Single Cross Sections 82
5.3.12.2 Multiple Cross Sections 83
5.3.13 Rudders 83
5.3.14 Thrusters 84

5.3.15 Structure Model ... 84
5.3.15.1 Compartments ... 84
5.3.16 Single Loading Condition ... 86
5.3.16.1 Mass Model ... 87
5.3.16.2 Second Order Surface Model ... 89
5.3.16.3 GZ-Curve ... 89
5.3.16.4 Offbody Points ... 90
5.3.16.5 Compartment Contents ... 90
5.3.16.6 Compartment points ... 92
5.3.16.7 Damping Matrix ... 92
5.3.16.8 Critical Damping Matrix ... 93
5.3.16.9 Restoring Matrix ... 93
5.3.16.10 Non-linear damping ... 94
5.3.16.11 Flooding Openings ... 94
5.3.17 Multiple Loading Conditions ... 95
5.3.18 Multi-body model ... 95
5.3.18.1 Multi-body stiffness matrix ... 96
5.4 Analysis ... 97
5.4.1 Hydrostatic Analysis ... 97
5.4.1.1 Single Stability Analysis ... 97
5.4.1.2 Multiple stability analysis ... 98
5.4.1.3 Wind Heeling Moment ... 99
5.4.1.4 Execute Stability Analysis ... 100
5.4.1.5 Hydrostatic Results Report ... 101
5.4.1.6 Hydrostatic Results Information ... 103
5.4.1.7 Results Report on File ... 104
5.4.1.8 Graphical Results ... 105
5.4.1.9 Hydrostatic Rule Checks ... 106
5.4.1.10 Damage Stability Analysis ... 108
5.4.1.11 AVCG (Allowable Vertical Centre of Gravity) analysis ... 108
5.4.2 Hydrodynamic Analysis by Wadam ... 110
5.4.2.1 Wadam Run ... 110
5.4.2.2 Execute Wadam analysis ... 120
5.4.2.3 Hydrodynamic Results ... 122
5.4.3 Hydrodynamic analysis by Wasim ... 123
5.4.3.1 Setup activity ... 123
5.4.3.2 Mass activity ... 127
5.4.3.3 Wasim activity ... 127
5.4.3.4 Using Morison models in Wasim ... 134
5.4.3.5 Handling models in mm in Wasim ... 134

1 INTRODUCTION

1.1 HydroD Overview

HydroD is an interactive application for computation of hydrostatics and stability, wave loads and motion response for ships and offshore structures. The wave loads and motions are computed by Wadam or Wasim in the Sesam suite of programs.

Wadam uses Morison's equation and first and second order 3D potential theory for the wave load calculations. Wasim uses Morison's equation and solves the 3D diffraction/radiation problem by a Rankine panel method. The incident wave potentials can be defined by Airy linear or Stokes 5^{th} order wave theory (Wasim only). Analysis can be performed in frequency domain or in time domain (Wasim only). The program generates the following results:

- Hydrostatic and stability computations for intact and damage condition
    - Hydrostatic data
    - Inertia properties
    - Righting moment
    - Wind heeling moment
    - GZ curve
    - Still water sectional loads
    - Analysis results checked against rules defined by internationally recognised codes
    - AVCG (Allowable Vertical Centre of Gravity) analysis
- Global response
    - First order wave excitation forces and moments
    - Second order wave excitation forces and moments (used to model springing effects, low frequency forces etc)
    - Hydrodynamic added mass and damping
    - First and second order rigid body motions
    - Sectional forces and moments
    - Steady drift forces and moments
    - Wave drift damping
    - Sectional load components (mass, added mass, damping and excitation forces)
    - Panel pressures
    - Fluid particle kinematics (for gap calculations and free surface animation)
    - Calculation of selected global responses of a multi-body system
- Transfer of structural loads to a finite element model
    - Inertia loads
    - Line loads on beam elements from Morison model
    - Point loads from pressure areas, anchor elements etc from Morison model
    - Pressure loads on plate/shell/solid elements
    - Internal tank pressure in compartments

Compartments are employed in hydrostatic and stability computations and they can be included in a mass model plus receive hydrostatic and hydrodynamic fluid pressure from a hydrodynamic run.

For details on the theory employed and calculation parameters in the wave load computation, the user is referred to the Wadam and Wasim user manuals.

1.2 HydroD Analysis Types

HydroD is mainly geared at the following analysis types:

1. Hydrostatic balancing
2. Stability analysis
3. AVCG analysis
4. Frequency domain analysis of floating rigid bodies.
5. Time domain analysis of floating rigid bodies.
6. Frequency domain analysis of fixed rigid bodies.
7. Deterministic analysis of fixed rigid bodies.
8. Time domain analysis of fixed rigid bodies.

1.3 HydroD in the Sesam System

HydroD is an integral part of the Sesam system. Finite element models (T*.FEM) generated in the Sesam programs GeniE, Patran-Pre and Presel are used as input to HydroD. Global response result interface files produced by HydroD (G*.SIF/SIN/SIU) can be read into e.g. Postresp for statistical post processing and viewing/printing of transfer functions etc. Loads interface files can be used in further analysis in Sestra. Displacements and loads interface files can be visualized and animated in Xtract.

1.4 How to read this Manual

The “Application Tools” chapter describes the common properties and tools of the application, independent of the specific data types used.

“Features of HydroD” gives an overview of the calculation results that HydroD is able to produce.

“User’s Guide to HydroD” gives information about how to model in HydroD.

“Dialog Description” gives a detailed listing of all the data input in HydroD.

Note that a lot of information is found from the tool tips inside the Graphical User Interface of HydroD.

1.5 Terminology and Notation

FEM: Finite element model

LMB: Left mouse button

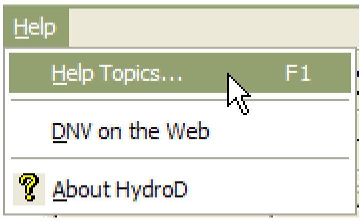
RMB: Right mouse button

1.6 Release Notes and Status List

There exists for HydroD (as for all other Sesam programs) information additional to this user manual. This may be:

- Reasons for update (new version)
- New features
- Errors found and corrected (bug fixes)
- Etc.

The main source for information on new features and bug fixes for HydroD is the Release Notes available through Help > Help Topics (F1) in HydroD.



This opens an HTML page. In the left browser pane of this HTML page the release notes and a command reference section can be accessed. The user manuals for HydroD, Wadam and Wasim are also available from this page.

Additionally, there are Status Lists for HydroD, Wadam and Wasim found on our website www.dnvsoftware.com. Go to services > support and click the Sesam Status Lists link and log into this service. Contact us for log-in information.

1.7 HydroD Extensions

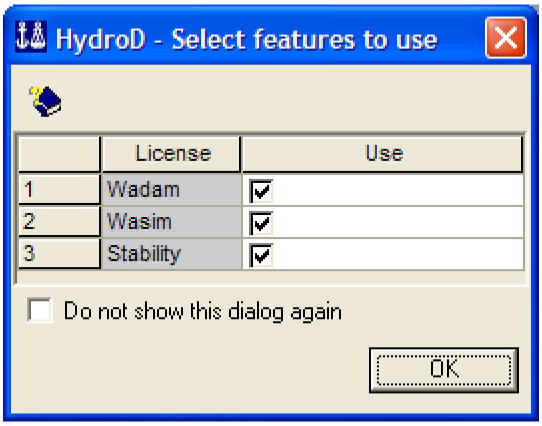
STAB Hydrostatic stability analysis – without this extension, the toolbars and menus for hydrostatic analysis will not be available.

2 APPLICATION TOOLS

This chapter provides an overview of the general application tools that HydroD provides the user with (independent of the data types).

2.1 Feature / license selection

When HydroD is started the following window appears.



In this dialog the user can select the tools that are to be used in the HydroD session. The user interface will then be configured such that only those folders, commands, tool buttons etc. that are relevant for the selected analysis type are shown.

Note that only components that have a valid license will be shown on the list. The Wadam option requires a “WADAM” license. The “Wasim” option requires a “WASIM_SOLVE” license. The Stability option requires a “HYDROD” license with a “STAB” extension.

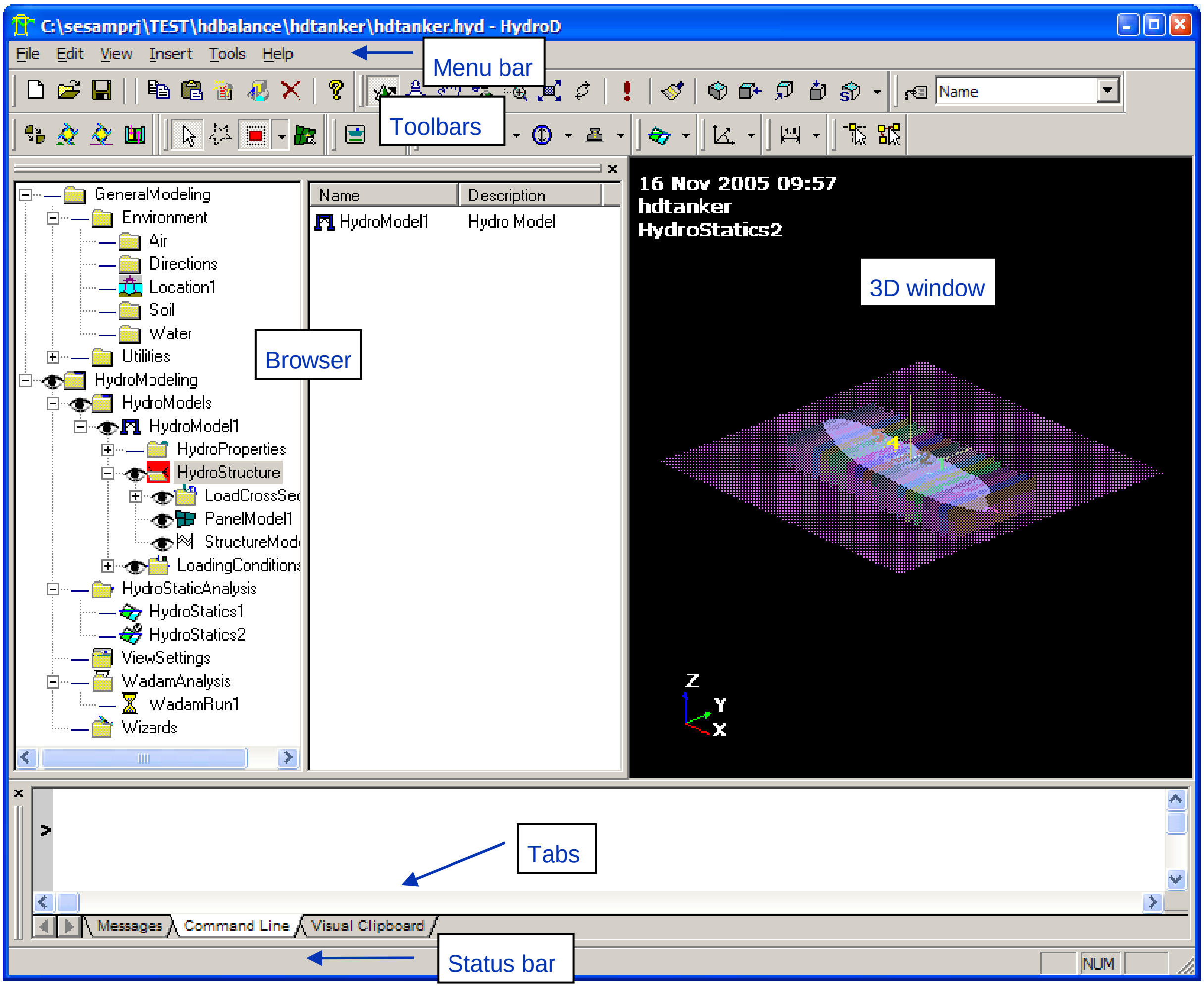
The selection of e.g. Wadam only will not prohibit that the workspace at a later stage can be extended to include Stability and Wasim analysis. The user just has to close HydroD and then restart the application and select the other analysis types in addition to Wadam.

Wadam and Wasim licenses are not taken if these options are selected. These licenses are only taken by the batch programs that are started from HydroD.

License option may also be given directly as a command line argument, e.g. when running HydroD in batch from a script file. More information is given in a later section on Batch execution.

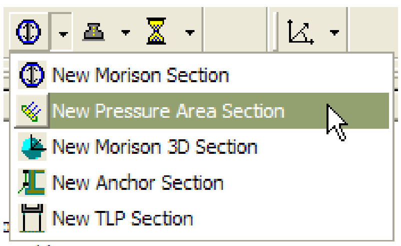
If the feature selection dialog has been turned off, it can be opened again by the “Edit | Select licenses / features…” command on the HydroD main menu.

2.2 Application Window



The application window contains six different areas:

- Menu area – Where users can access the menu actions with the mouse button or through pressing Alt key followed by the relevant underlined shortcut letters. Pressing Alt+F+N in sequence will for instance activate the new workspace dialog. Pressing Alt+F+W+1 opens the workspace you worked on in the previous session. Note that the relevant tool tip hint for the selected menu item appears in the status bar.
- Browser area – Where users can access the objects loaded and generated. Objects are set visible/invisible by clicking on the eye in the browser.
- Toolbar area – By pressing the left mouse button on a toolbar entry, the user has direct access to different actions. When moving the mouse cursor over a toolbar entry, the relevant tool tip will pop up in the status bar. Note that many of the toolbar entries are of pull-down type. This means that each button contains many entries.
  Change to a different entry by pressing the arrow to the right of the toolbar. Note that you have to press the button after you have selected it.



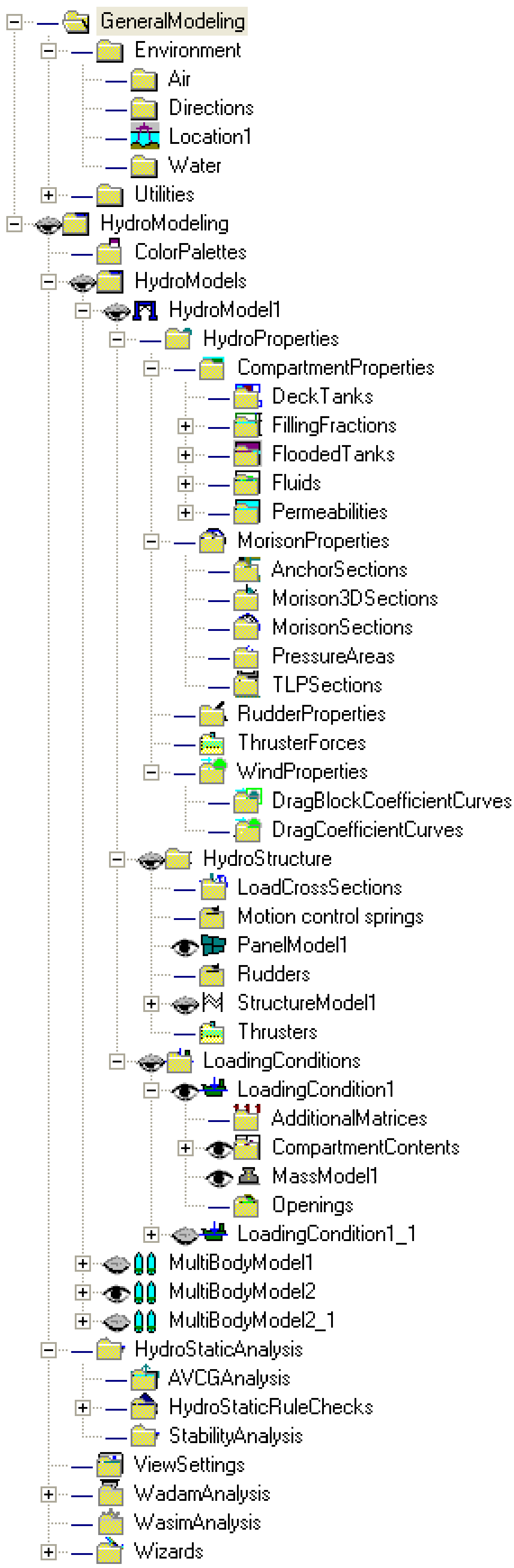
- 3D window area – Where objects are visualized.
- Tabs area – Contains three different tabs for program messages (such as warnings, errors and information messages), command line arguments (scripting commands) and visual clipboard (extraction of coordinates, distances etc).
- Note that most dialog windows may be enlarged by stretching the window frames

2.3 Data Organization

Data read into and generated in HydroD is stored in a hierarchical tree structure.

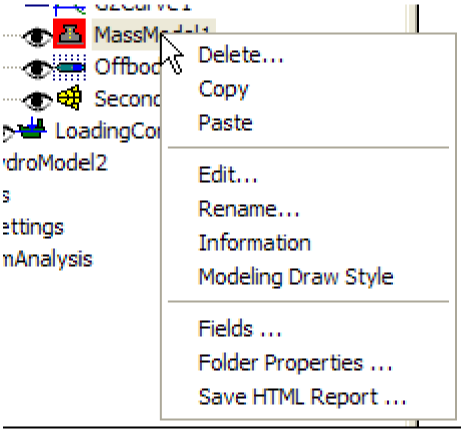
The main folder structure is as follows:

- General Modelling
    - Environment: Data related to wind, water, air and soil
    - Utilities: Utility geometry (guide planes, guide points, guide curves etc)
- Hydro Modelling
    - Hydro models: Structural data of different bodies
    - Hydrostatic Analysis: Hydrostatic and stability analyses
    - View settings: Instances of view settings in the 3D modelling window
    - Wadam analysis: Execution parameters for Wadam
    - Wasim analysis: Execution parameters fro Wasim
    - Wizards: Wizards guiding the user through the relevant definition dialogs

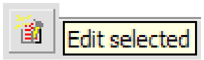

GeneralModeling
Environment
Air
Directions
Location1
Water
Utilities
HydroModeling
ColorPalettes
HydroModels
HydroModel1
HydroProperties
CompartmentProperties
DeckTanks
FillingFractions
FloodedTanks
Fluids
Permeabilities
MorisonProperties
AnchorSections
Morison3DSections
MorisonSections
PressureAreas
TLPSections
RudderProperties
ThrusterForces
WindProperties
DragBlockCoefficientCurves
DragCoefficientCurves
HydroStructure
LoadCrossSections
Motion control springs
PanelModel1
Rudders
StructureModel1
Thrusters
LoadingConditions
LoadingCondition1
AdditionalMatrices
CompartmentContents
MassModel1
Openings
LoadingCondition1_1
MultiBodyModel1
MultiBodyModel2
MultiBodyModel2_1
HydroStaticAnalysis
AVCGAnalysis
HydroStaticRuleChecks
StabilityAnalysis
ViewSettings
WadamAnalysis
WasimAnalysis
Wizards

2.4 Context Sensitive Actions

Very often the most convenient and easiest way of reaching an action is to right click the relevant object. This activates the menu belonging to the selected object/objects. If you for instance right click a mass model, you get access to all relevant actions and properties of the selected mass model:

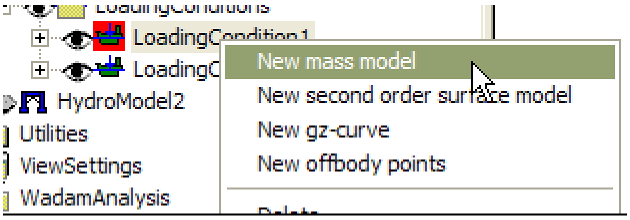


The Edit action, which is typically used to change the information already defined, may also be accessed from the toolbar:



The Edit action is also entered if you double-click on the object.

In order to create new objects, one may right click the relevant parent folder to get access to the definition dialog:



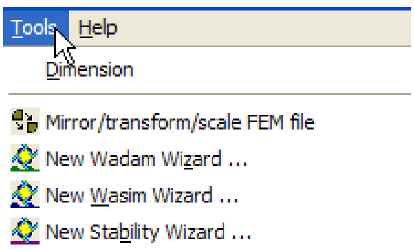
It is also possible to reach the same menus by right clicking the objects in the 3D window.

2.5 Wizards

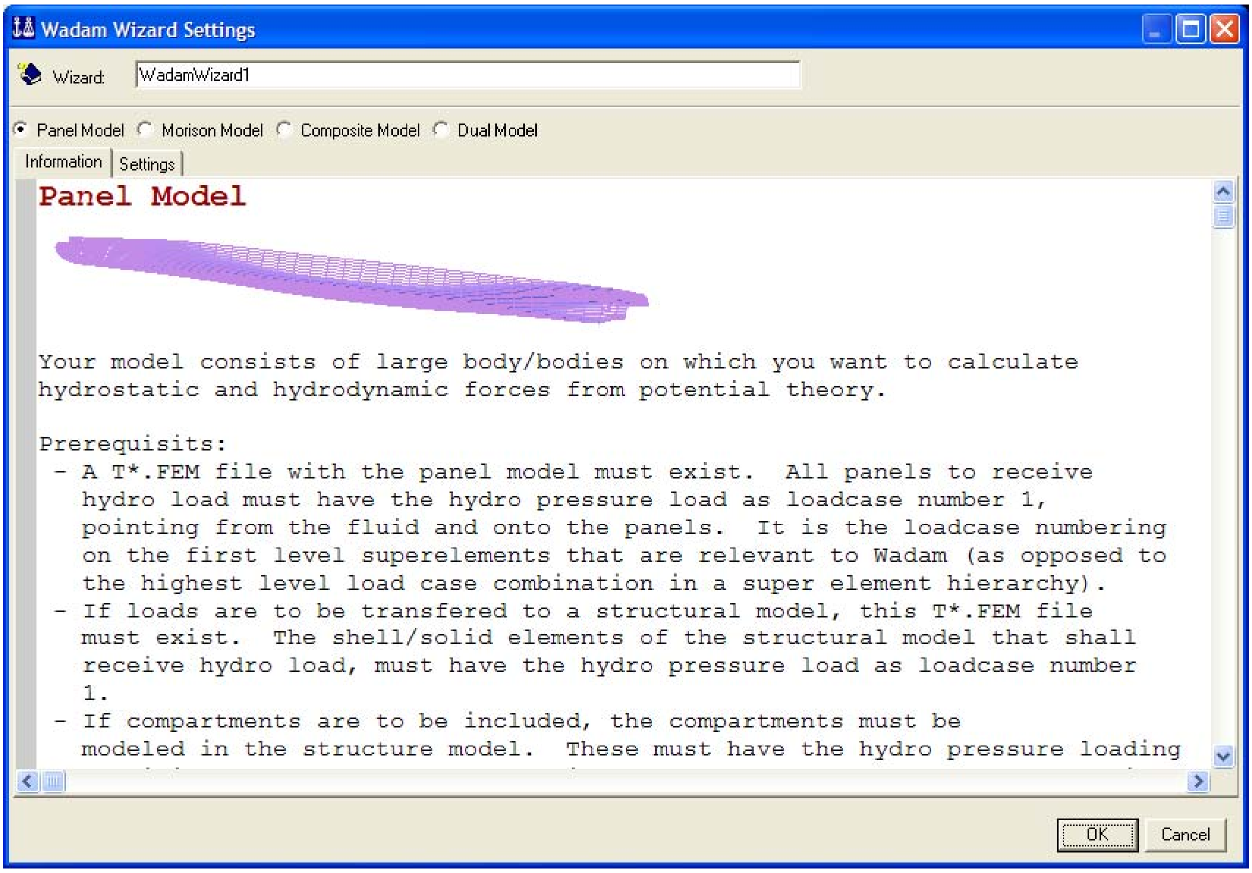
There are three wizards available.

- Stability wizard
- Wadam wizard
- Wasim wizard

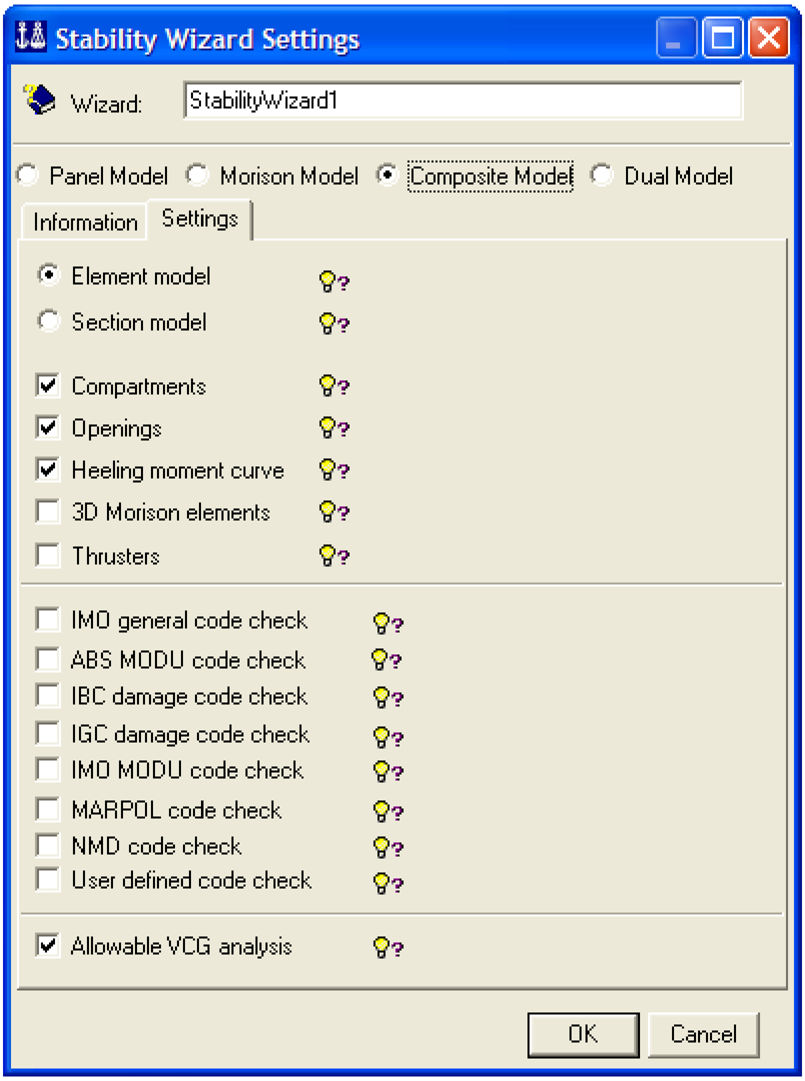
They will guide the user through the necessary steps to set up a specific analysis, and are highly recommended for both new and experienced users. The wizards can be accessed through the toolbar buttons  and menu item:



The wizards start by making you select between the four main model configurations that you may use (these will later be explained):

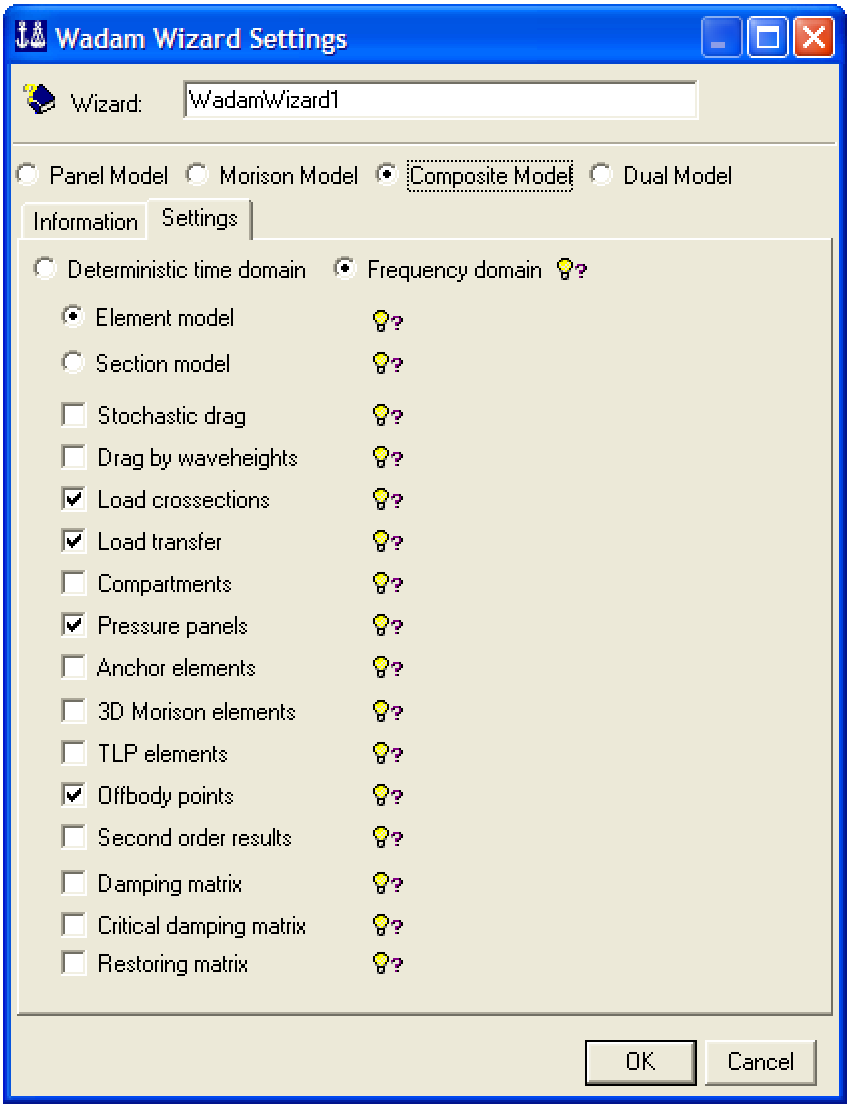


For each model type you get two dialog tabs. The first one (“Information”) contains general information about the model configuration you have selected. The second tab (“Settings”) is where you select the components that you want to include in your analysis, here shown for the Stability wizard, the Wadam wizard and the Wasim wizard respectively:

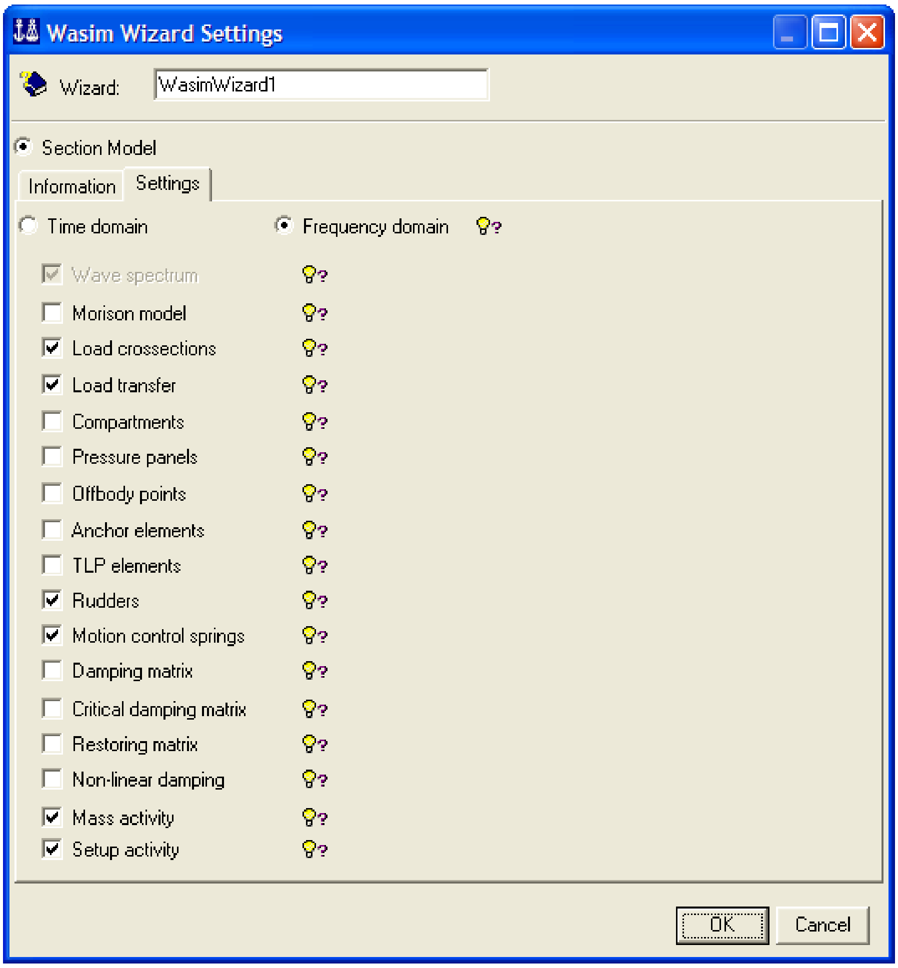


The selection of Element model means that the panel model is imported into HydroD in the form of a Sesam Interface File (T-file).

The alternative selection of Section model, means that the external geometry is imported into (or created in) HydroD in the form of a set of 3-diemnsional curves. The panel model is then generated inside HydroD from the Section model.

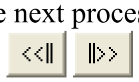


A multi-body analysis cannot be defined completely from the Wadam wizard, some manual definitions are required.

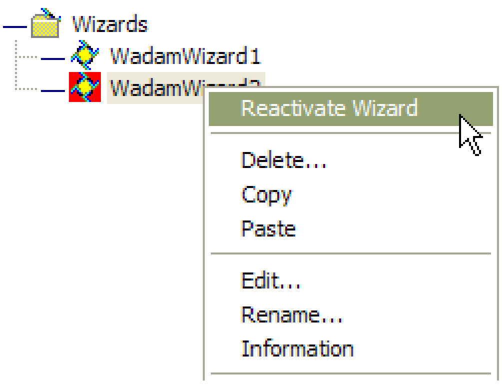


For Wasim it is not possible to import a panel model. Only the Section model option is allowed.

When the "OK" button has been pressed, the wizard panel will appear:

Each time you press "Next step", the next process dialog in your wizard will pop up. Use the  buttons to step to previous/future dialogs in the process. If for instance you have previously generated environment data that you want to reuse, you would typically step past all the environment dialogs.

Wizards can be reactivated at a later stage by right clicking them and choosing "Reactivate":



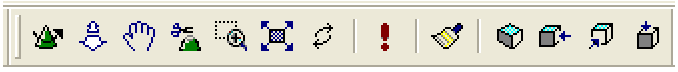
2.6 Graphical Interaction

RMB and LMB are abbreviations for right and left mouse buttons.

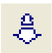
The following shortcut keys are always available:

- Pan (move the visible image) in the 3D window by pressing Ctrl+RMB
- Zoom by pressing Shift+RMB
- Zoom rubber band by pressing Ctrl+Shift+RMB
- Rotate by pressing Alt+RMB

In addition there are a number of graphical interaction buttons on the toolbar menu:



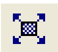
 When toggled, rotate the visible image with the RMB

 When toggled, zoom the visible image in/out with the RMB

 When toggled, pan the visible image with the RMB

 When toggled, the clipping plane closest to the viewer can be moved by pressing the RMB and dragging the cursor up/down. This will enable the user to see inside cuts of the models.

 Press this button and make a rubber band selection with the RMB.

 Zoom entire visible image inside the window.

 When toggled, the model will continuously rotate.

 When pressed, all graphical objects will be redrawn.

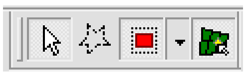
 Toggle on/off colour coding of properties

 Rotate the model such that you view it from point [+ x, - y, - z] and zoom all.

 Rotate the model such that you view it from the x/-y/z-axis and zoom all.

2.7 Selection

The selection toolbar looks as follows:

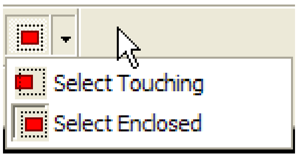


Objects can only be selected in the 3D window when  is toggled.

By pressing LMB on objects in the 3D window you will then be able to select them. Keep Shift pressed to select multiple objects (continue a selection).

Multiple objects can also be selected with a rubber band or polygon selection. Rubber band selection is done by moving the mouse cursor to a point outside all visible objects, pressing the LMB and keeping it pressed until the desired objects are enclosed by the rubber band.

Similarly when  is pressed you can select objects enclosed by a polygon. The polygon is defined by pressing the left mouse button on points in the 3D view and ending the polygon in the same point it started.



The next entry on the selection toolbar specifies whether you want to select all objects completely enclosed by the rubber band/polygon, or all objects that intersect the area inside the rubber band/polygon.

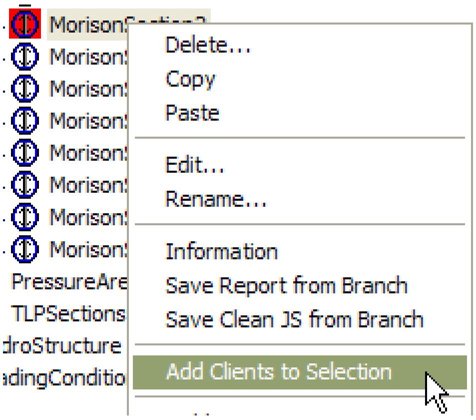
Objects can also be selected in the browser, similar to the way that files are selected in Windows Explorer. Click the LMB on the relevant object. Continue a selection (in the right browser pane) by keeping Ctrl pressed. Select a range (in the right browser pane) by keeping Shift pressed.

A selection is cleared by pressing the LMB at a point in the 3D window where there are no objects.

When  is pressed, you can select points on the panel model and the Morison model to verify coordinates, display dimensions etc.

2.7.1 Add Clients to Selection

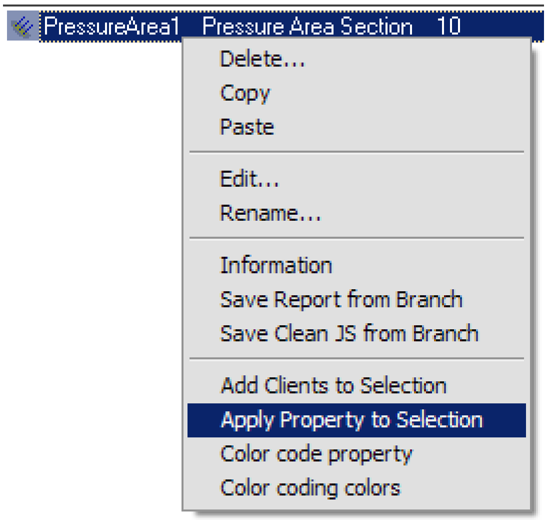
By right clicking a property object (like Morison section, pressure area section, TLP section etc) you can reach the “Add Clients to Selection” option:



This will add all the objects that use the selected property to the selection. If these objects are visualized in the 3D window, they will be highlighted with the selection colour. In other words, “Add Clients to Selection” is an easy way to see who uses a given property.

2.7.2 Apply Property to Selection

By right clicking some of the properties (pressure area sections, TLP sections, anchor sections etc), you can get access to the “Apply Section to Selection” action.

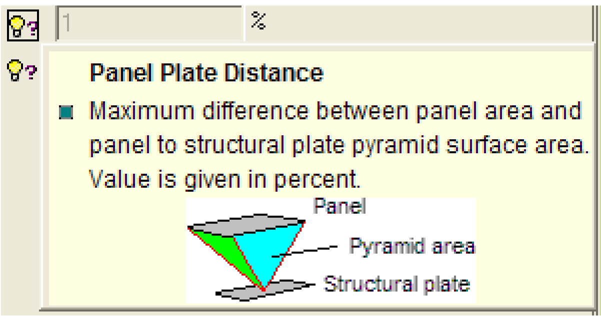


This will result in an attempt to apply the right clicked property to all the objects that are currently selected. In other words, this enables one to select objects in the 3D window or in the browser and subsequently set a given property on them.

Selected objects of a type that can not accept such a property (a Morison3d section can for instance not be applied to an anchor element) will simply be skipped (i.e. it doesn’t matter if the selection contains other objects than the ones to which you want to apply the property as long as these objects are of a different type).

To maintain the selection while reaching the “Apply Section to Selection” action, either keep shift pressed or directly right click the relevant property (as opposed to left clicking it first to select it, and then right clicking it to reach the action).

2.8 Tool tips



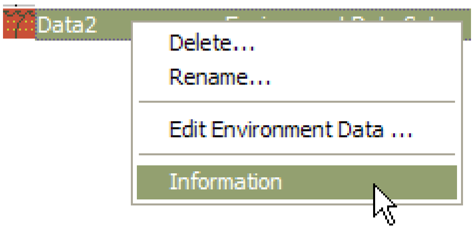
All dialogs are packed with tool tips that pop up when the user positions the mouse cursor over them. There are two main types:

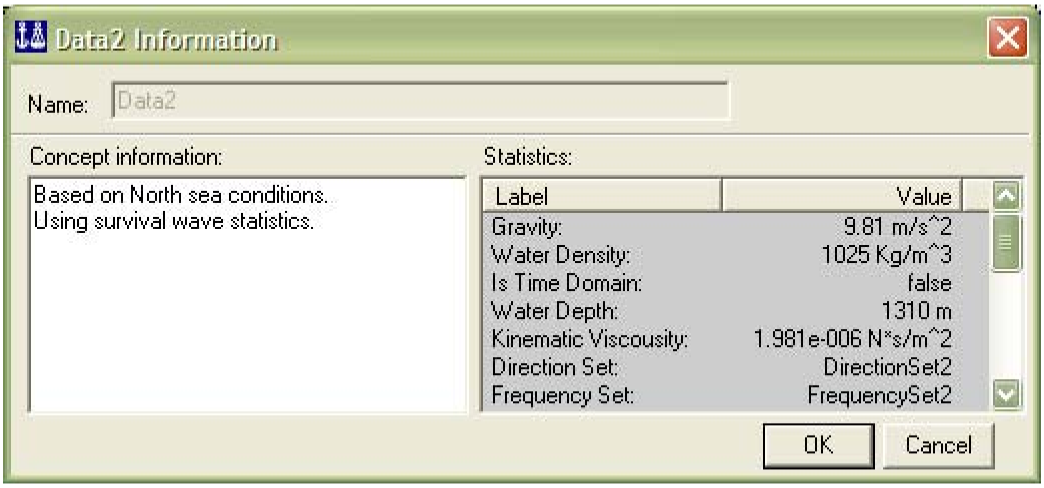
1.  Provides general information of the work process/purpose of a dialog.
2.  Provides specific information related to a single control in the dialog.

The tool tips are essential in the modelling process. Many issues covered in the tool tips are not included in this manual.

2.9 Reporting

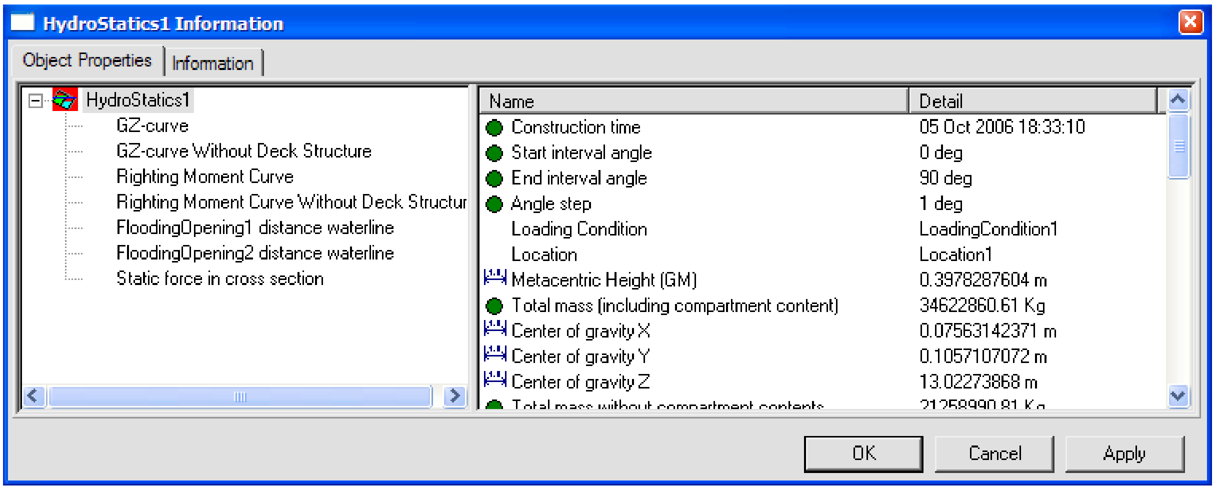
When right clicking (pressing the RMB) on an object in the browser you can choose “Information”:

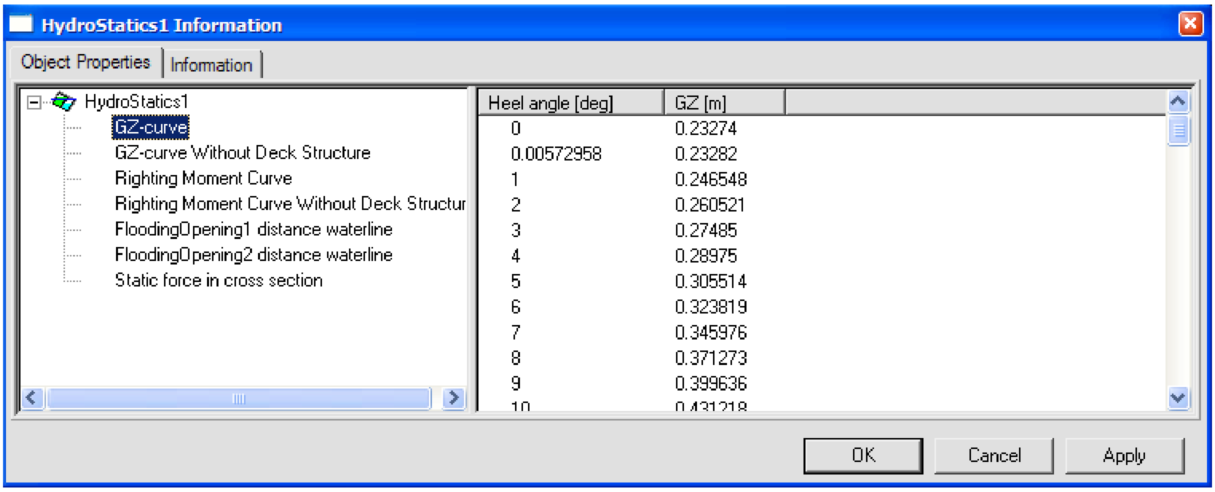




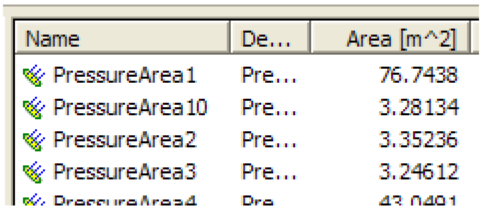
This will list relevant information and statistics from the object. In addition the user can write comments that are stored together with the object. This can for instance be used to provide information about governing assumptions and modelling methods.

For some objects, the information is given in form of sub folders in the Information window:

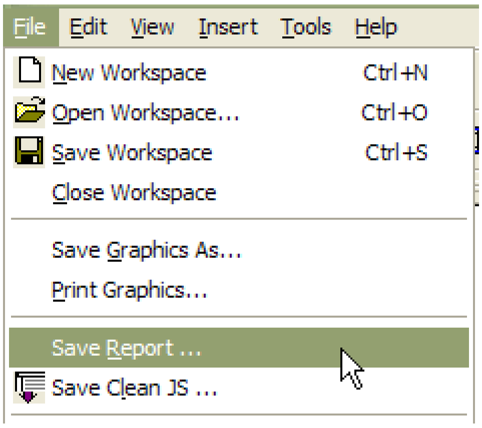


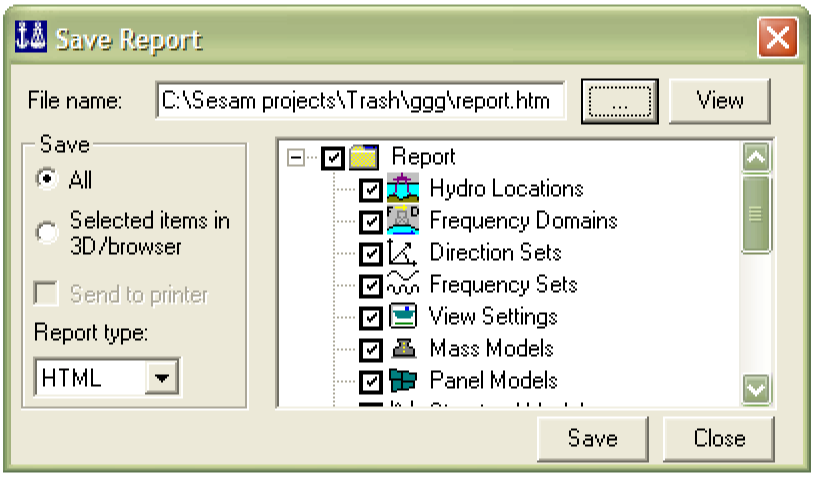


In the browser, some of the defining characteristics of the objects are listed:



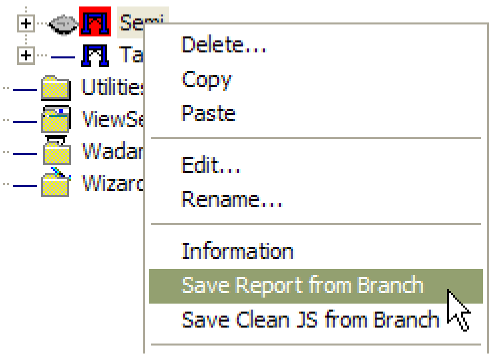
From the File menu you can access the save report dialog:





Here you select the relevant data type and file name and generate a listing of HTML or XML format (for Word or Excel).

The same dialog can also be reached by right clicking some of the objects:



The report will then only be generated from the selected branch.

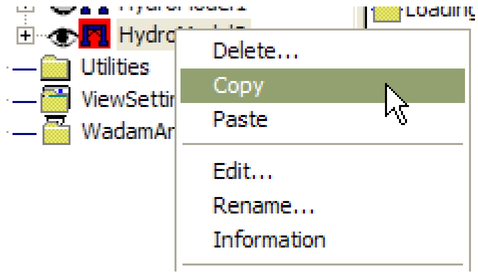
2.10 Copy/Paste

Copy/paste can be done on all objects. This option can be very useful when almost similar cases shall be tested. Copy/paste is done in the following manner:

- Clear the selection (press the left mouse button at a point in the 3D window where there are no visible objects).
- Select the objects that you want to copy either from the browser or in the 3D window.
- Press Ctrl+C or use the copy toolbar button 
- Clear the selection.
- Position yourself in the browser folder that you want to copy the selected objects into.
- Press Ctrl+V or use the paste toolbar button 

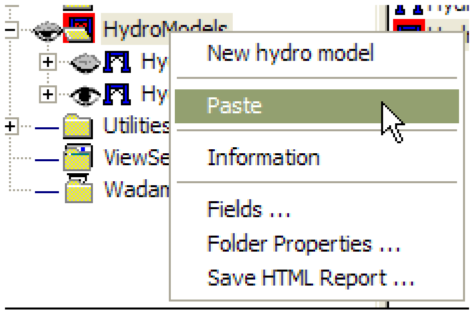
Alternatively you can do the following:

- Right click the relevant object in the browser and choose copy.



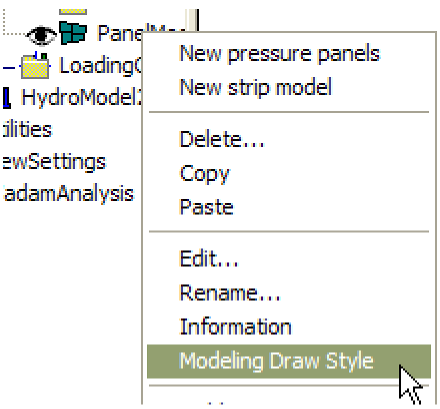
- Right click the relevant destination folder in the browser and choose paste.

An entire hydro model can for instance be copied by right clicking it in the browser and selecting copy, right clicking the hydro models folder and selecting paste.

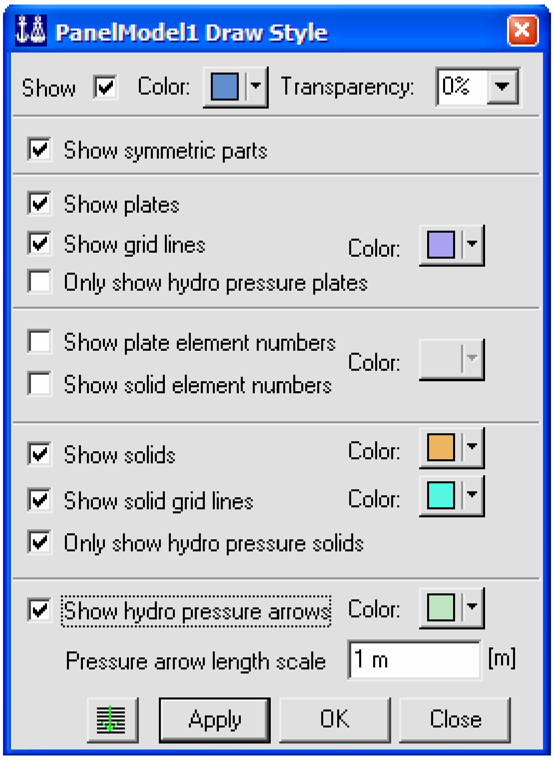


2.11 Draw style Settings

All objects that can be visualized in the 3D window have a "Modelling Draw Style". This can be accessed by right clicking the object, by use of the toolbar button , or from the toolbar/edit menu.



Here you specify how (if at all) an object shall be visualized in the 3D window. For a panel model the "Modelling Draw Style" dialog will for instance look like:



All "Modelling Draw Style" dialogs have the  button.

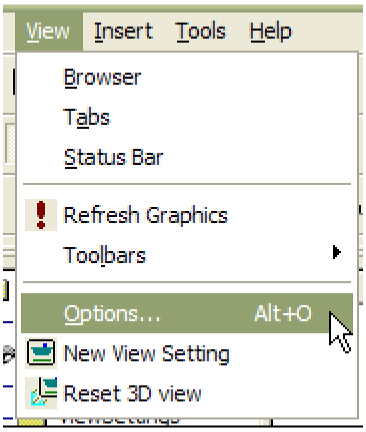
When this is pressed, the settings specified in the dialog will be applied to all selected objects of the same type. Note that colours are not affected.

The visible/invisible setting, "Show", is also directly available from the eye in the browser.

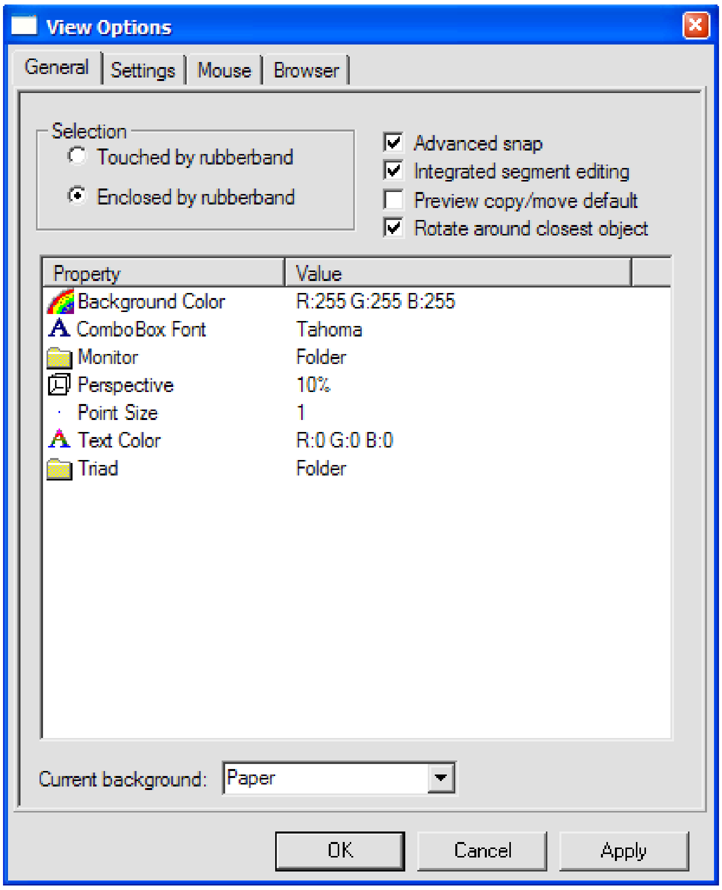
One should note that it is possible (and sometimes necessary) to access the draw style settings of objects while a process dialog is up. For instance when defining TLP-elements, one may be interested in seeing node numbers on the Morison model. You will then have to open the "Modelling Draw Style" dialog of the Morison model and select that you want to see node numbers.

2.12 View Options

From the “View” menu you can reach the view options dialog:

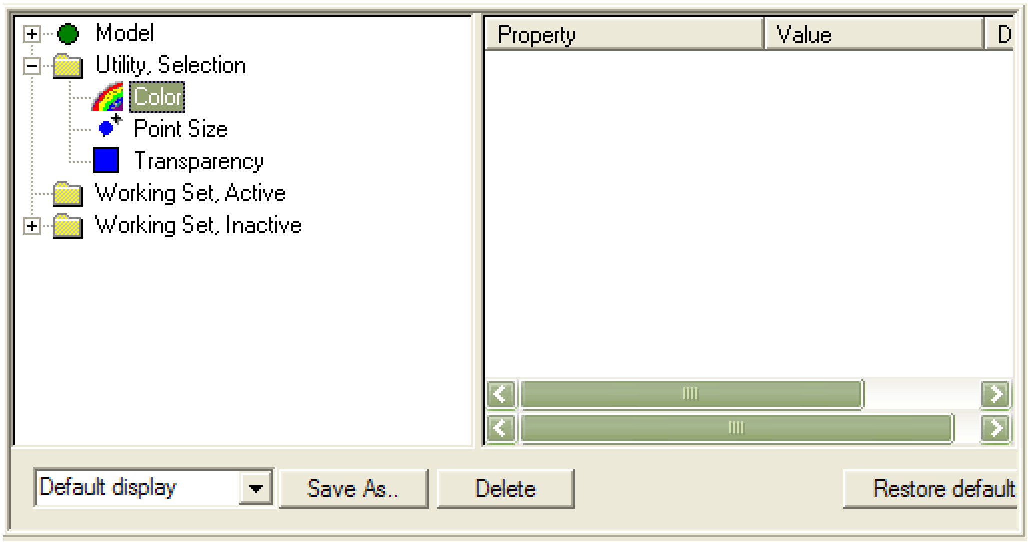


This contains several tabs. The first tab, “General”, looks as follows:



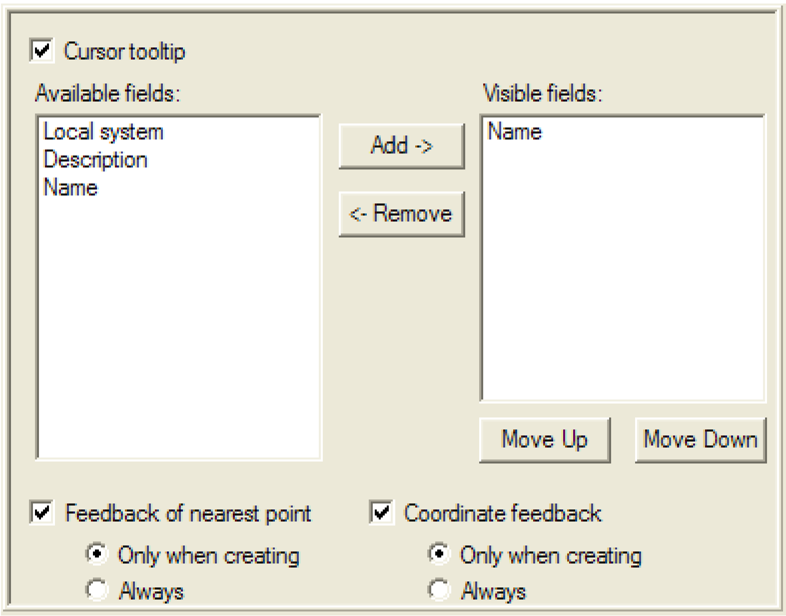
Here you specify some general settings for the 3D window. Double click the entries in the list to change their value.

The second tab, “Settings”, looks like:

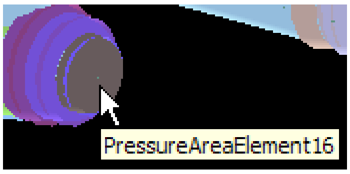


Here you specify how selected objects shall be marked (selection colour etc) as well as the draw style settings of some system objects. Double click the entries in the list to change their value.

The third tab, “Mouse”, looks like:



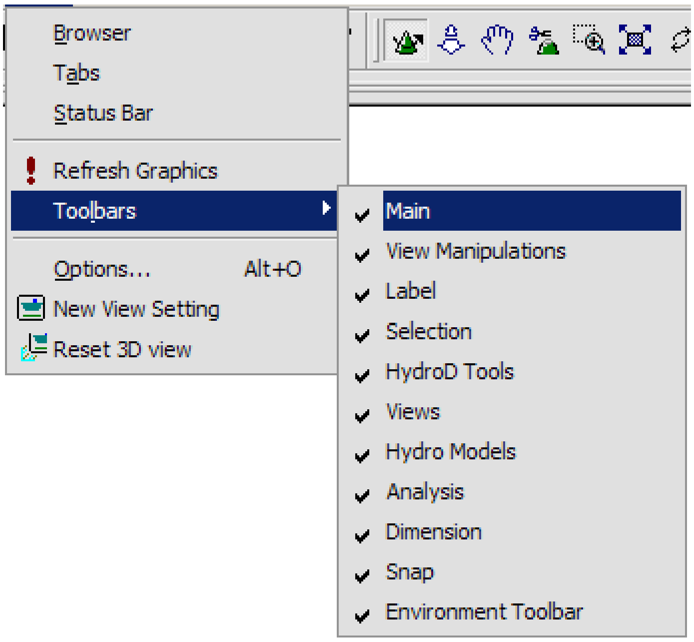
When moving the mouse cursor over an object in the 3D window and letting it rest there for a second or two, a tool tip will pop up if “Cursor tool tip” is toggled.



This may contain name, description and/or local coordinate system (if relevant) based on what you have selected in “Visible fields”. At the bottom of this tab you may specify that you want the closest point and or coordinate highlighted as you move the cursor over the model. The coordinate feedback appears in the status bar (bottom left corner).

2.13 View Toolbars

There are a number of predefined toolbars. These may be turned on/off from the View menu:

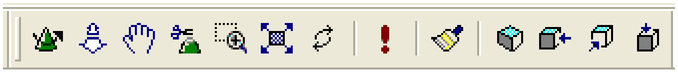


2.13.1 The Main Toolbar

This toolbar may be used for main actions, like File – New/Open, Save workspace, copy/paste etc.

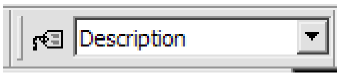
2.13.2 The View Manipulations

This toolbar is used for graphical interaction.



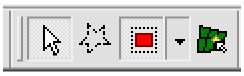
2.13.3 The Label Toolbar

This toolbar may be used to label entities in the graphics.



2.13.4 The Selection Toolbar

This toolbar is used to customize the selection feature.

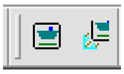


2.13.5 The HydroD Tools Toolbar

This toolbar is used to activate some tools, like the Wizards, transformation of T#.fem file, and Compartments hydrostatic balancing

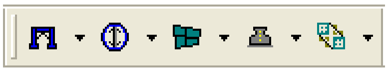
2.13.6 The Views Toolbar

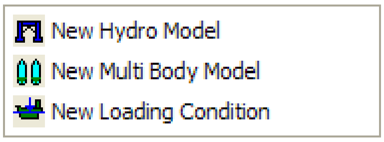
This toolbar is used for View settings.

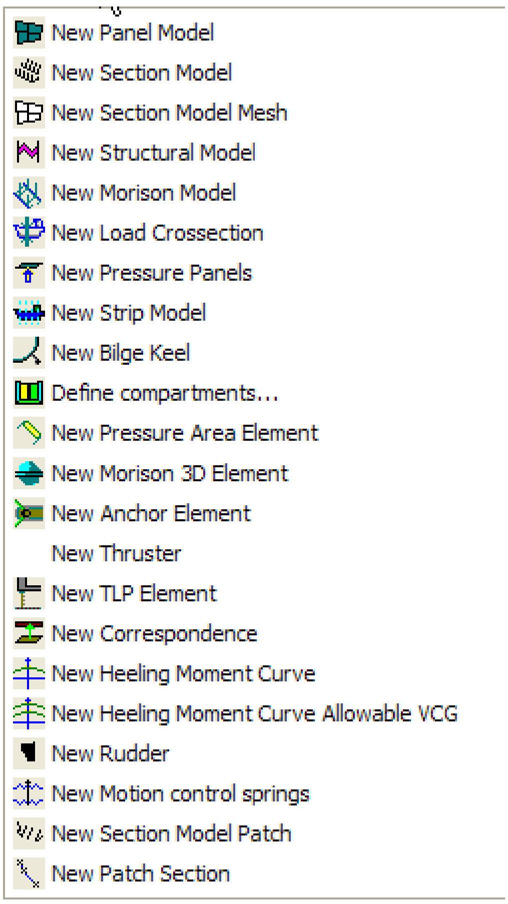


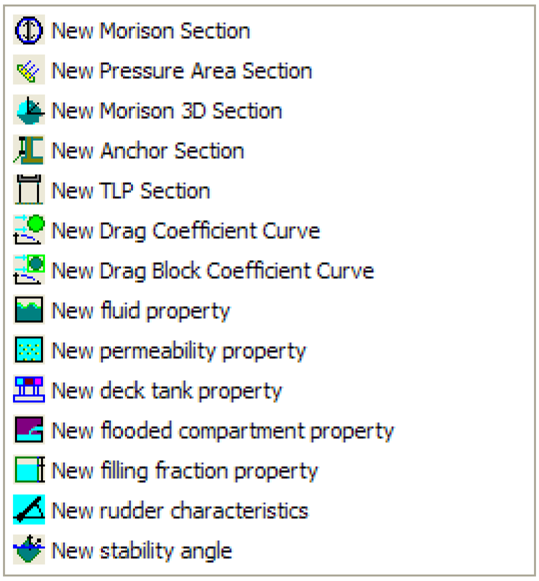
2.13.7 The Hydro Models Toolbar

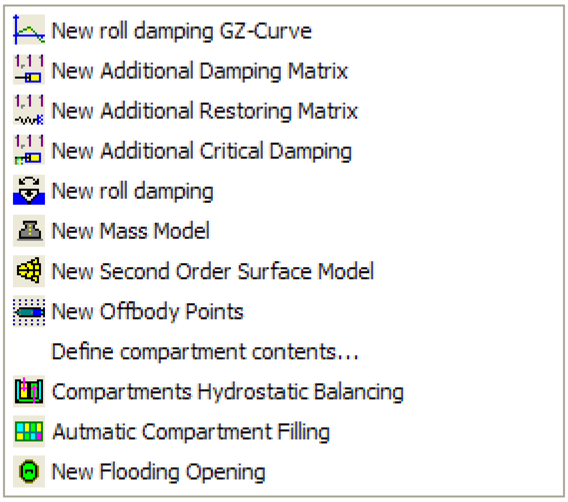
This toolbar is of the pull down type toolbar, used to define the necessary hydro models and their properties, like panel model, Morison mode, mass model etc.





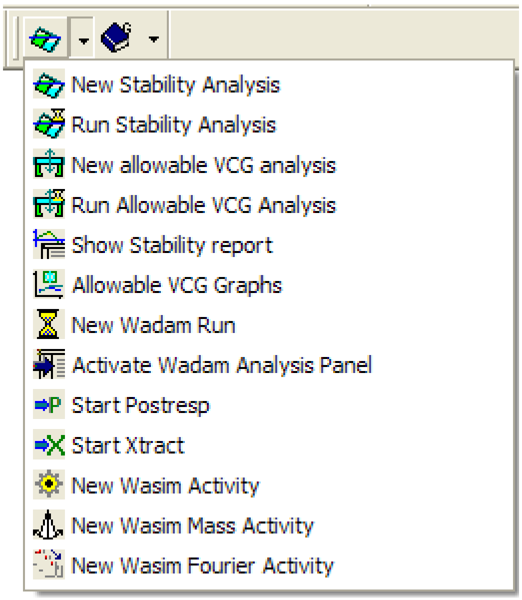


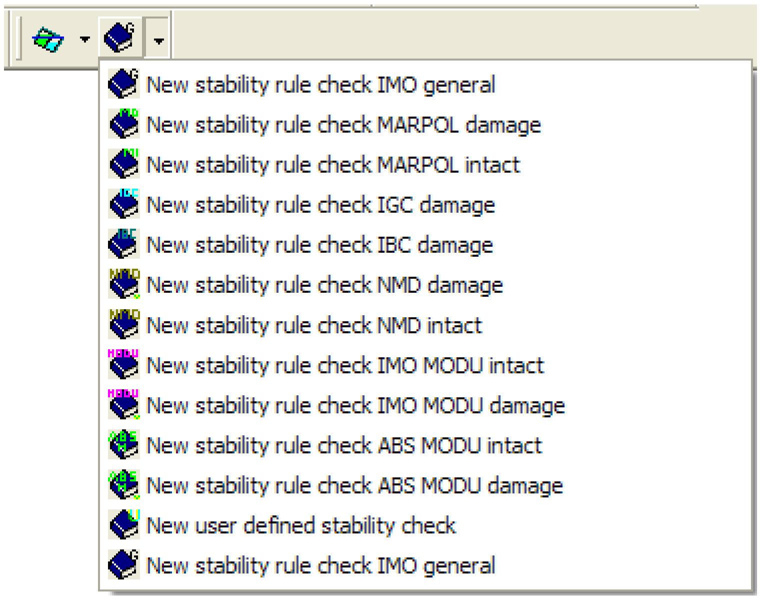




2.13.8 The Analysis Toolbar

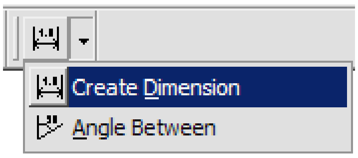
This toolbar is used to define & start the analyses, run stability rule checks and also to start post-processing.





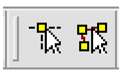
2.13.9 The Dimension Toolbar

This toolbar is used to draw values of dimension and angle for specified parts of the structure.



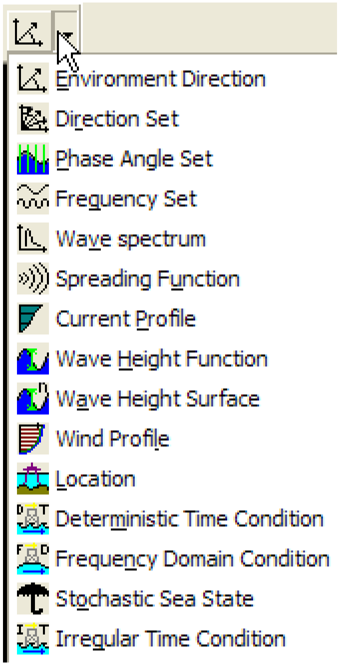
2.13.10The Snap Toolbar

This toolbar controls how to snap on to existing points etc.



2.13.11 The Environment Toolbar

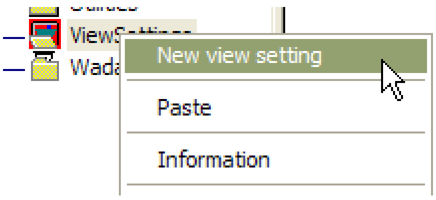
This toolbar is used for definition of environmental data, like wave periods, frequencies etc.



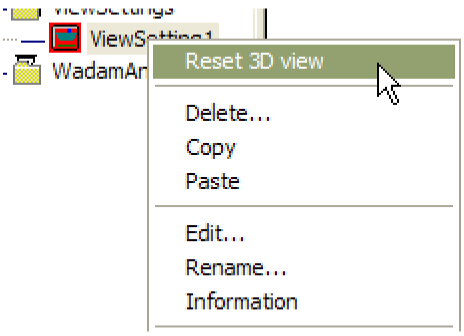
2.14 View Settings

When you have visualized a set of objects in a certain way, and you want to store these settings for future use, you can generate a view settings object. Later you can activate this object to reset the 3D view exactly as it was (including the view point).

The relevant dialog is accessed by pressing the button or right clicking the view settings folder:



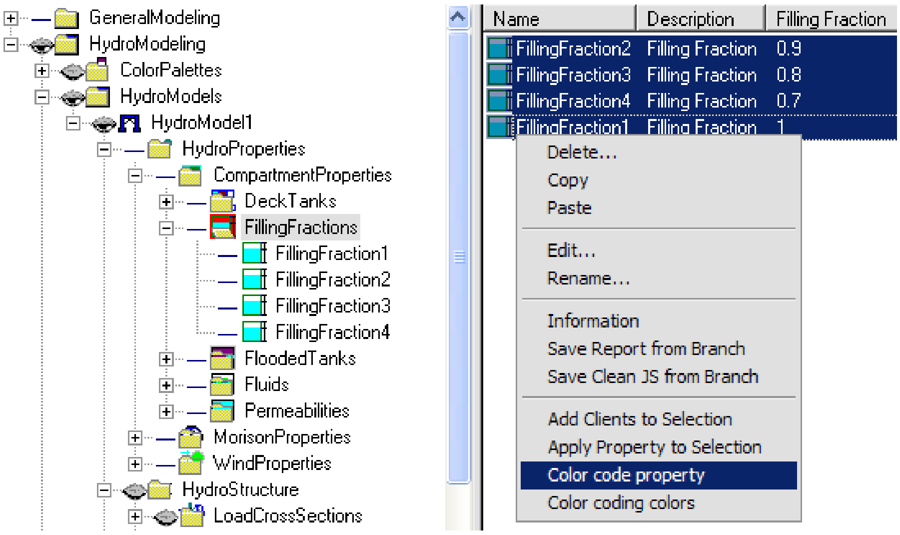
To reactivate the view setting folder press the button or right click the view setting folder:



2.15 Colour Coding of Properties

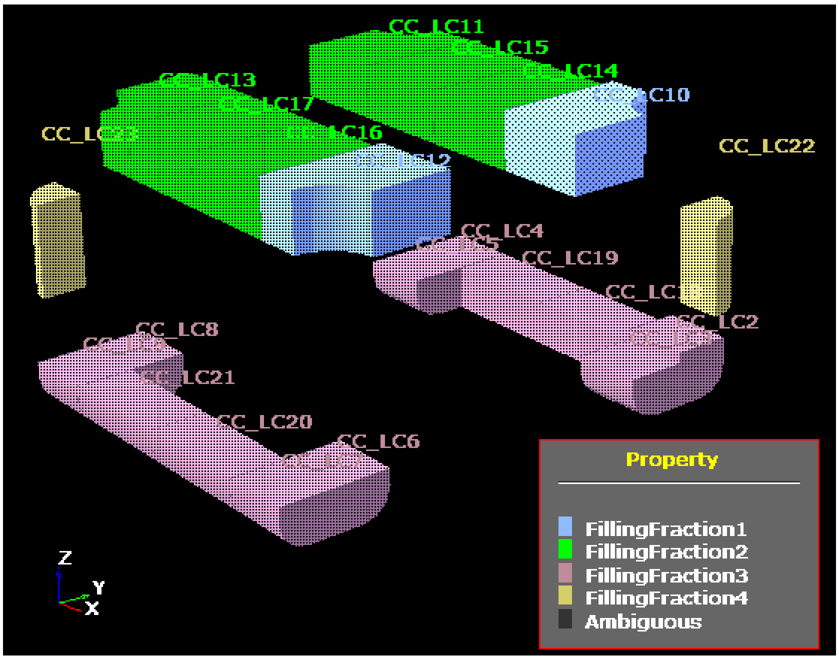
All properties may be displayed in separate colours for verification purposes.

This is enabled by right-clicking the applicable property sets and selecting “Colour Code Property”:

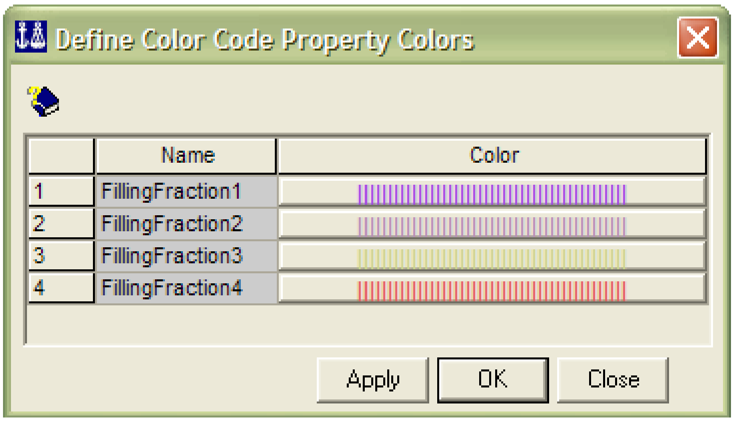


In order to colour all the relevant properties, all sets should be selected; in this example all 4 Filling Fractions are selected. The resulting display is shown below, including a colour legend.

The colours may be toggled on/off by clicking the Colour Code button. 



The colours used for colour coding can be changed through the “Colour coding colours” dialog reached by right clicking one of the properties:

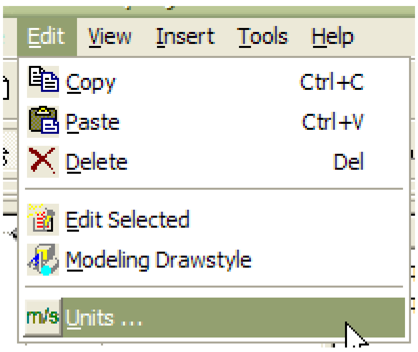


By clicking on one of the entries in the right column, the standard colour selection dialog pops up:

2.16 Units

All data can be input with a unit specification:

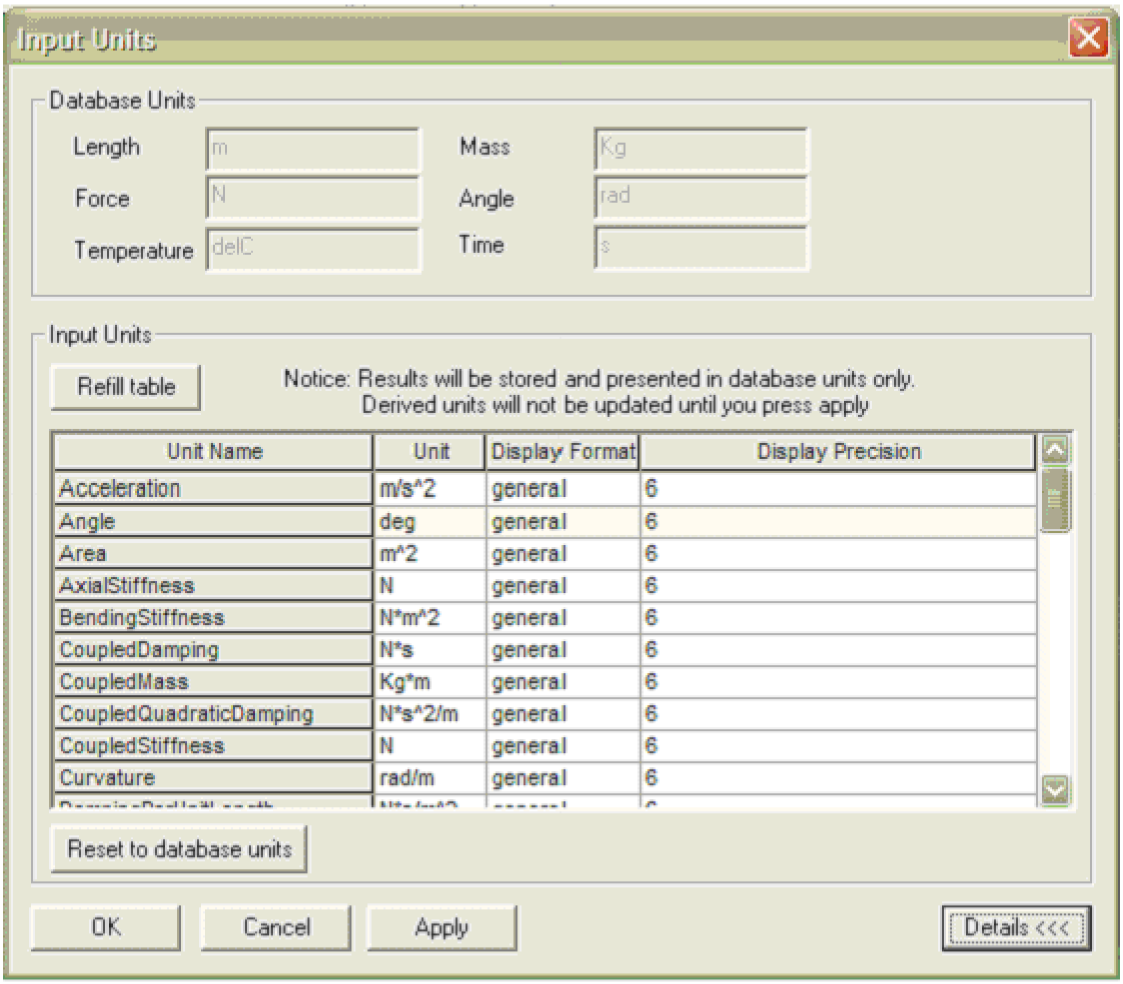
The database units (the units in which the data are stored) can ONLY be set when a new workspace is established (and not changed later). This may however be read initially from an input command file (.js file).



Default display units (the units that are assumed when no explicit unit is specified by the user in the input) can be set at any time. The relevant dialog is accessed through the "Edit" menu:

Note that the database units are effectively defined when creating the model(s) in a pre-processor.

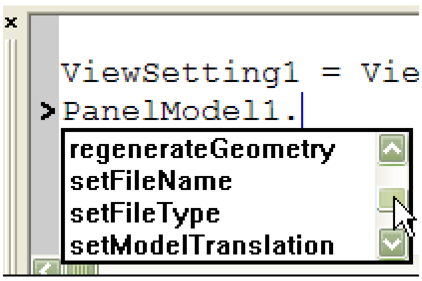
It may look like this (after the "Details" button has been pressed):

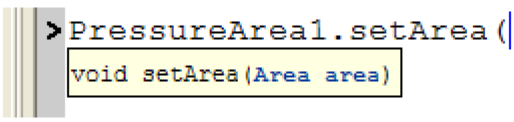


Here you specify the desired unit, the display format (fixed, scientific or general) and the display precision.

2.17 Scripting

All user input automatically generate the corresponding scripting commands, and users can choose to give their input solely as scripting. The scripting/command line window gives the user a list of available commands on given objects (press Tab to use this and Esc to remove it):





It also shows the argument list of the selected function.

The scripting language (Jscript) has support for many advanced options like loops, user defined functions, if statements etc. Further information may be found on http://msdn.microsoft.com/. More help on the commands is available through Help > Help Topics (F1) in HydroD.

The Help information can also be reached by writing the following in the command line window: document("my_directory");

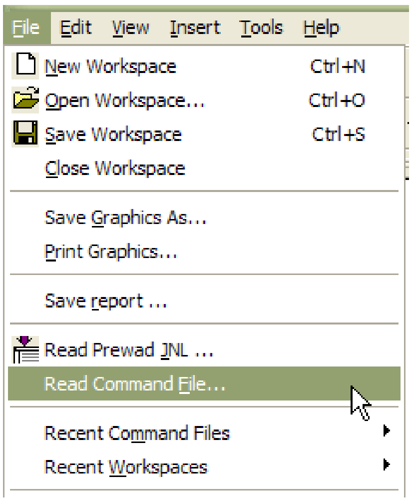
my_directory may for instance look like: `c:/HydroD/doc/`

This will produce an html hierarchy in the directory "my_directory". Double click index.html in this directory to see a documentation of all available scripting commands in the application.

Experienced Sesam users may be used to select/define many objects of a certain type with loop commands like GROUP (where you define a start and stop value and a step). In HydroD the user would instead employ a loop. To define a frequency set with periods ranging from 2 to 40 seconds with step of 2 seconds, the user could for instance write:

```
FrequencySet4 = FrequencySet(FrequencyTypePeriod,Array(2));
for (period = 4; period <= 40; period += 2)
  FrequencySet4.addPeriod(period);
```

One should however note that this type of commands cannot be written directly into the command line window in HydroD (because this window interprets one line at a time, and does not understand multi-line commands like the loop statement). Instead the user will either have to write all commands in a scripting file (*.js file) and load this file into HydroD:

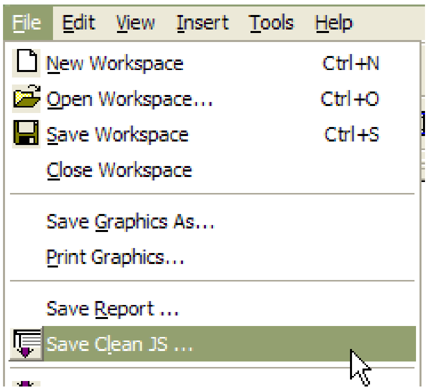


Alternatively one may write the connected commands (like the two for loop lines) in an editor (like Notepad), and then copy and paste them into the command line window in HydroD.

One may also note that it is possible to zoom (Shift+RMB) and pan (Ctrl+RMB) in all edit windows (including the scripting and messages windows).

2.18 Save Clean JS

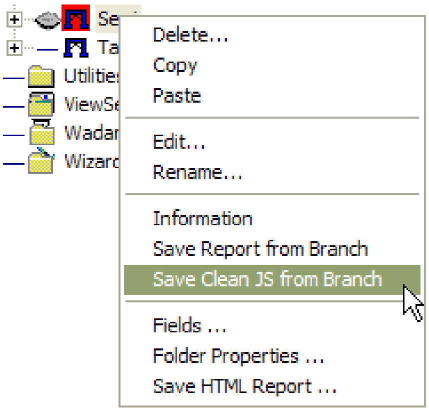
From the “File” menu the “Save Clean JS” command can be reached:



This will produce a script file containing all the objects in your workspace in a defined sequence.

This file is very useful to rebuild the model, while the automatic script (log) file will typically be more difficult to use, because it will contain all changes made by the user etc.

It is recommended to save the Clean JS file in the same folder as your original input files (*.FEM) – then you will get relative path names for the input files.



The same command can also be reached by right clicking some of the folders:

Only the objects under the selected branch will then be written out to the scripting file.

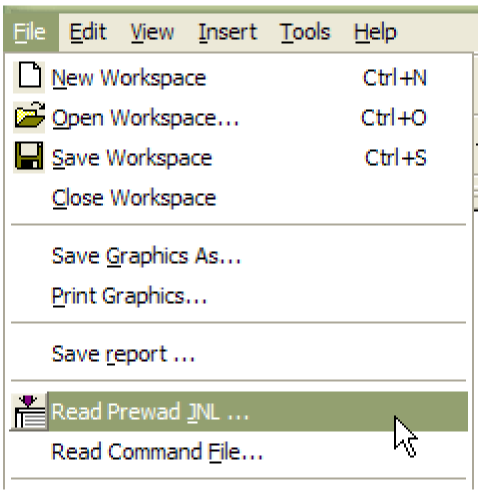
Note that the run analysis command (execute Wadam) will not be included.

2.19 Batch

HydroD can be run in batch from the command line (with a scripting file). Write “HydroD /?” (alternatively the full path to HydroD if HydroD is not included in the environment variable “PATH”) in a command window to see how the command arguments should be given.

2.20 Prewad JNL Files

Prewad JNL (journal) files can be interpreted and read into HydroD. The relevant dialog is reached from the file menu:



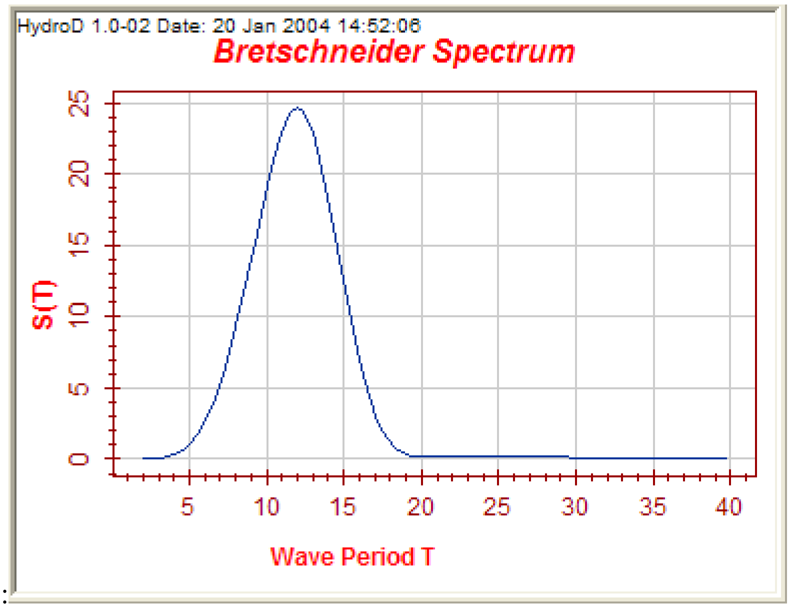
Prewad is the old pre-processor for Wadam.

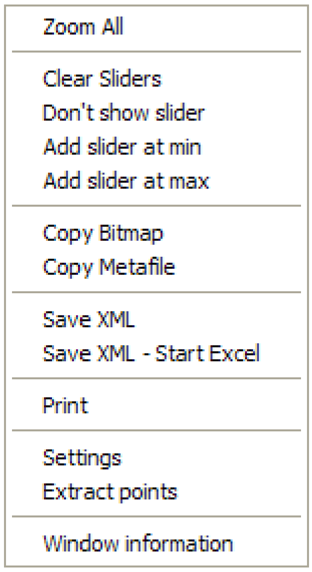
One should note that some object types cannot be directly mapped from the Prewad.jnl file and into HydroD objects, and it may sometimes be impossible to do the reinterpretation. This will of course be reported by the program during the read process.

A Prewad journal file for a multi-body analysis cannot be read into HydroD.

2.21 Graph Controls

Many of the dialogs utilize a graph control



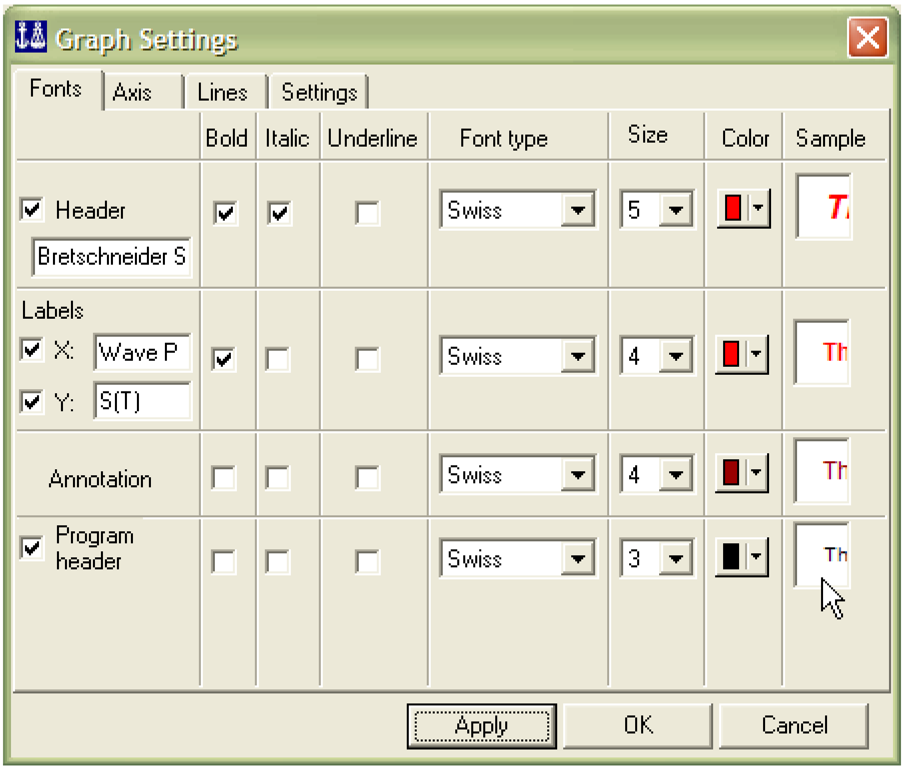


:

The slider will only follow one of the curves in the graph. To change from one curve to another click on the curve that is to be followed by the slider while pressing the Ctrl-button.

Zoom in the graph control by pressing Shift+RMB and pan with Ctrl+Shift+RMB.

By right clicking inside the graph control, one gets access to the graph control settings (as well as the other functionality listed):



In the graph control settings headers, font types, colours, axis types, line types etc can be specified.

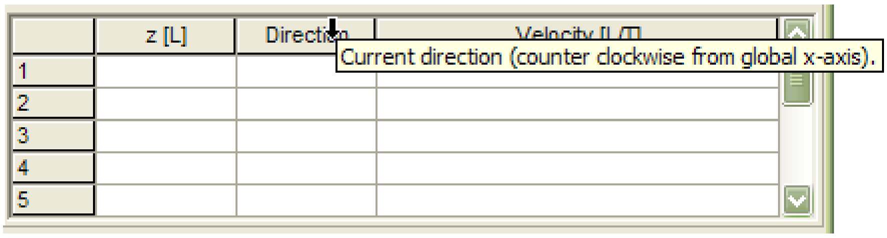
The option “Extract points” can be used to print a specific value for a point on the x axis.

2.22 Grid Controls

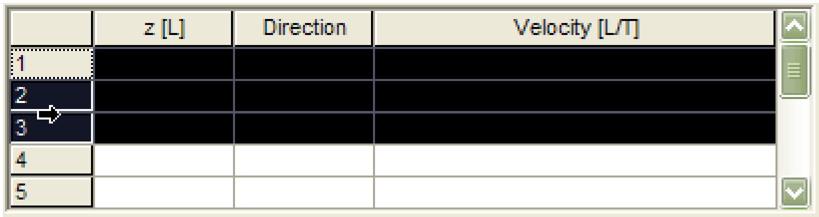
The grid controls are used in many dialogs.

Data can be copied from Microsoft Excel and pasted into such controls (press Ctrl-V to paste).

By moving the mouse over the first row, useful hints on what is defined in each column will appear:



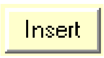
Multiple rows are selected by dragging the mouse cursor along the first column, keeping left mouse button pressed. Press Ctrl-C to copy the content of selected cells:



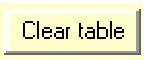
Columns are added by positioning the mouse cursor in the last row and pressing the down arrow.

Some dialogs that contain a table also have the buttons described below. This functionality makes it easier to enter and modify table values:

- 

 If the values have a systematic variation Fill table may be used. Enter first, last and step values and click the Fill table button. Values are automatically filled in the table.
- 

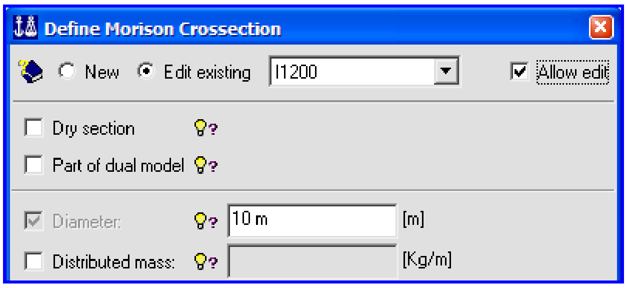
 The Insert button is used to insert one or more values. Number of values inserted equals number of rows selected. Values are inserted before first selected row and are calculated based on linear interpolation.
- 

 The Remove button removes all selected rows.
- 

 The Clear table button clears the table.

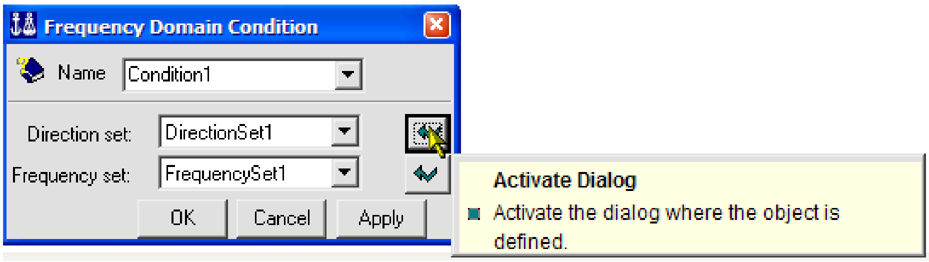
2.23 Dialog New/Edit Functionality

Many dialogs have a special functionality for handling both creation of a new data object and editing an existing. When the dialog is opened in 'new-mode' the New and Allow edit options are automatically ticked and the dialog is ready to accept data. When opened in 'edit-mode' the Edit existing option is ticked. All data fields are disabled and the dialog is in a read-only state. To open for editing the Allow edit option must be ticked.



2.24 Late Definition Functionality

Some dialogs offer what may be called 'late definition functionality'. Such functionality is found in dialogs that include other data objects. An example is the Frequency Domain Condition dialog where dialogs for defining direction set and frequency set objects may be activated directly from this dialog.



3 FEATURES OF HYDROD

This chapter gives an overview of the results that HydroD can produce.

3.1 Hydrostatic Analysis

Hydrostatic and stability computations may be run for both intact and damage conditions. HydroD will compute the draught and heel/trim angles to ensure equilibrium. Compartments may be flooded or balanced by HydroD. A wind heeling moment, calculated by HydroD, may be included.

The results of this computation are found in the Report window. Analysis results may be checked against rules defined by internationally recognised codes.

3.1.1 Hydrostatic Data

Given in the information tab of the report window, in save report or by right clicking an object and choosing “Information”:

- Displaced volume
- Mass with and without compartment fluid
- Centre of gravity and centre of buoyancy
- Centre of flotation
- Metacenter
- Trim moment
- Compartment information

3.1.2 GZ curve

HydroD computes the GZ curve for the structure (with and without the influence of deck compartments for offshore structures).

- The GZ curve is displayed
- The shortest distance between a flooding opening and the sea surface is displayed
- Zero crossings of the GZ-curve are reported
- Zero crossings of the lowest flooding opening are reported
- Change in trim and waterline is reported at each heel angle
- Integrals of the GZ-curve are reported

Some of the characteristics of the GZ-curve computation are listed below:

- The elements are cut in the waterline/free surface to give exact volume and mass computation
- Exact level of free surface in compartments is computed at every heel angle

3.1.3 Righting Moment

HydroD computes the righting moment.

- The righting moment curve is displayed
- Righting moment zero crossings are reported
- Integral of righting moment is reported

3.1.4 Wind Heeling Moment

HydroD computes the wind heeling moment based on a drag coefficient curve and a drag block coefficient curve.

- The wind area is computed at every heel angle
- The heeling moment curve is displayed
- Integral of heeling moment is computed

3.1.5 Openings

Flooding openings in the hull may be defined at selected locations.

- The distance from the opening to the waterline is displayed
- Heeling angle of intersection with the waterline is printed
- An opening may be connected to a compartment, making this flooded when submerged

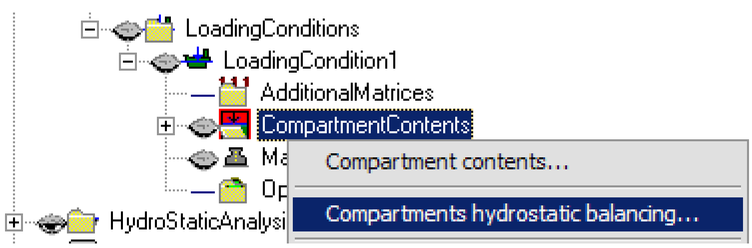
3.1.6 Cross Section Data

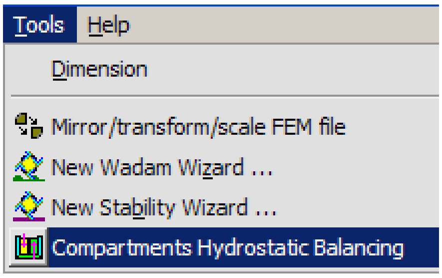
HydroD computes the sectional loads in the still water condition on the specified side of the defined cross sections.

- Cross sectional loads, forces and moments, may be both displayed, e.g. longitudinal bending moment, and printed
- The results may be split into components from mass and buoyancy separately

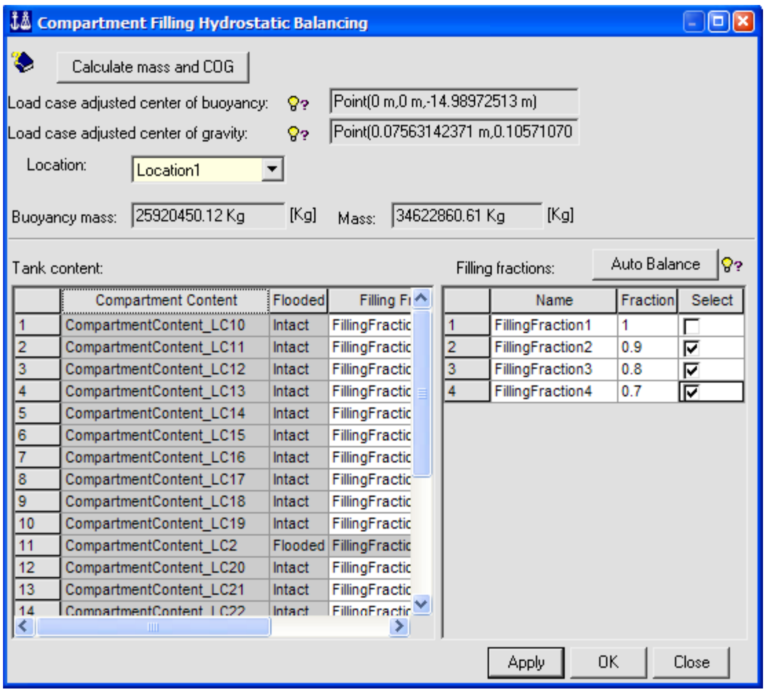
3.1.7 Hydrostatic Balancing of Compartments

HydroD may calculate the filling ratio of compartments necessary to obtain equilibrium of gravity and buoyancy forces. This is started either from the browser or from the Tools menu:





The following dialog window controls this calculation:

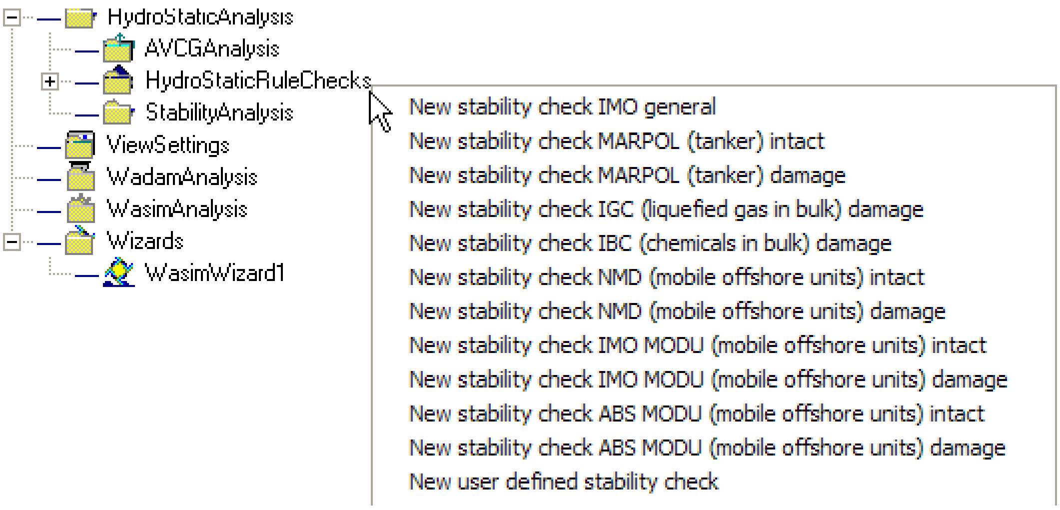


At least three Filling Fractions (compartment properties) must be selected in the table.

HydroD may also compute the required trim, heel and draft to balance a given mass distribution. This issue is discussed in the chapter about loading conditions.

3.1.8 Hydrostatic Rule Checks

HydroD may check the analysis results against rules defined by internationally recognised codes, here shown as defined from the browser:



In addition to the predefined code checks there is also an option for a user defined code check.

3.2 Hydrodynamic Analysis

The hydrodynamic analysis is performed by the programs Wadam and Wasim. For details on the theory employed, calculation parameters in the wave load computation and available results, the user is referred to the Wadam/Wasim user manuals.

3.2.1 Hydrostatic Data

On the Wadam print file (WADAM1.LIS) the following hydrostatic results can be found:

- Displaced volume
- Water plane area
- Centre of buoyancy
- Transverse and longitudinal metacentric height
- Heave-heave, heave-roll, heave-pitch, roll-roll, pitch-pitch and roll-pitch restoring coefficients
- Stillwater sectional loads

In a load transfer analysis detailed hydrostatic loading including panel pressures, distributed beam loads, tension leg tether and anchor mooring pre-tension loads are included on the loads interface files.

In a Wasim analysis the same data can be found on the LIS file from the Wasim_Solve activity.

3.2.2 Inertia Properties

On the Wadam print file (WADAM1.LIS) and the Wasim_Solve LIS-file the following inertia properties are reported:

- Total mass
- Centre of gravity
- Roll, pitch and yaw radius of gyration
- Roll-pitch, roll-yaw and pitch-yaw centrifugal moments

In a load transfer analysis acceleration cards are written to the loads interface files (see "Loads Interface paragraph" paragraph). For beam elements (Wadam only) added mass from Morison theory may be included as mass or distributed load. If the Morison model is used as the mass model, mass loads (as opposed to just the acceleration) are transferred to the loads interface file.

3.2.3 Global Response

Global response results from Wadam are reported on the Wadam print file (as non-dimensional results) and on the hydrodynamic results interface file, the G-file (see "Hydrodynamic Results Interface File" paragraph), may include:

- Transfer functions of first order wave excitation forces and moments
- Transfer functions of second order wave excitation forces and moments
- Transfer functions of first order rigid body motion
- Second order transfer functions of second order rigid body motion
- Deterministic motions at specified phase angles in time domain
- Transfer functions of sectional forces and moments
- Transfer functions of pressures on selected panels
- Transfer functions of fluid particle kinematics (pressure and velocity) at specified points
- Added mass matrices
- Damping matrices
- Restoring matrix
- Steady drift forces and moments
- Wave drift damping
- Sectional load components including mass, added mass, potential damping and excitation forces.
- Eigen Solutions
- Calculation of selected global responses of a multi-body system

Global response results from a Wasim frequency domain analysis are printed on output files (*.fmout and *.ldout) from the Fourier activity, and on the hydrodynamic results interface file, the G-file (see "Hydrodynamic Results Interface File" paragraph) These data may include:

- Transfer functions of first order wave excitation forces and moments (fixed vessel analysis)
- Transfer functions of first order rigid body motion (free motion analysis)
- Transfer functions of sectional forces and moments
- Transfer functions of pressures on selected panels
- Added mass matrices (forced motion analysis)
- Damping matrices (forced motion analysis)
- Relative motion at selected points

Time histories of the global results from a Wasim time domain analysis are written on output files from the Solve activity. These time histories can be displayed in HydroD.

3.2.4 Load Transfer

In a load transfer analysis with Wadam the following results are written to the loads interface file:

- Nodal loads from pressure area, TLP, anchor and 3D Morison elements
- Distributed beam loads from Morison model
- Pressure loads (including diffraction and added mass) on plate/solid elements
- Rigid body acceleration
- Time invariant current profile forces from Morison model

On fixed structures in time domain loads may be calculated up to the linearized free surface.

On the Wadam print file (WADAM1.LIS) load sums for the transferred loads are printed, in addition to information on the structural model and matching information of the load transfer.

In a load transfer analysis with Wasim the following results are written to the loads interface file:

- Nodal loads from pressure area, TLP and anchor elements
- Distributed beam loads from Morison model
- Pressure loads (including diffraction and added mass) on plate/solid elements
- Rigid body acceleration

From a frequency domain analysis these are the RAOs for the accelerations and pressures (as with Wadam). From a time domain analysis they are snapshots of these quantities taken at selected points in time. In this case the pressure is the total pressure, i.e. the hydrostatic component is included.

On the Wasim_Stru print file information on the structural model and matching information of the load transfer is found

3.2.5 Other Results

The following results from a Wadam analysis are available for output in Wamit's internal format:

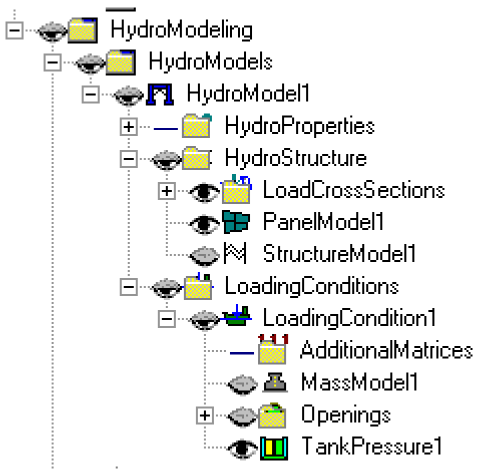
- Second order panel pressures
- Second order pressure in fluid
- Second order wave elevation

3.3 3D Visualization

HydroD has a powerful visualization engine. All data input by the user and read into HydroD is visualized in the 3D window, giving the user an opportunity to visually quality check the model.

3.3.1 Display Control

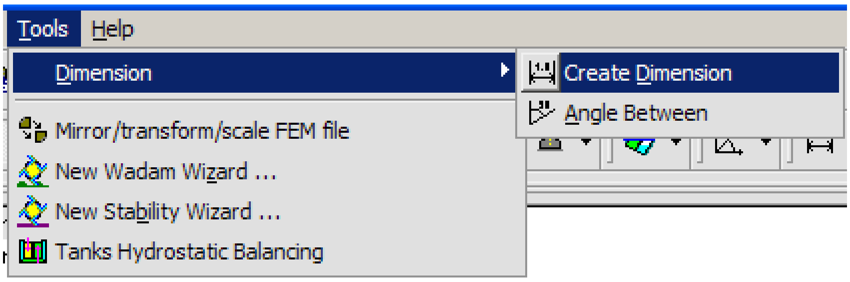
The information displayed in the 3D window is easily controlled by opening and closing the eye in the browser. This is done by clicking on the eye.



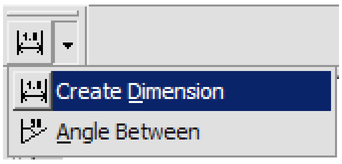
Similar control can be achieved by use of the Modelling Draw Style feature, as described in section Draw Style Settings.

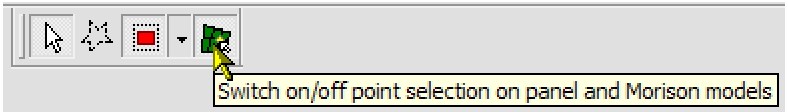
3.3.2 Dimension

The dimensions of the model can be checked by use of the Dimension tool. This is activated from the Tools menu,

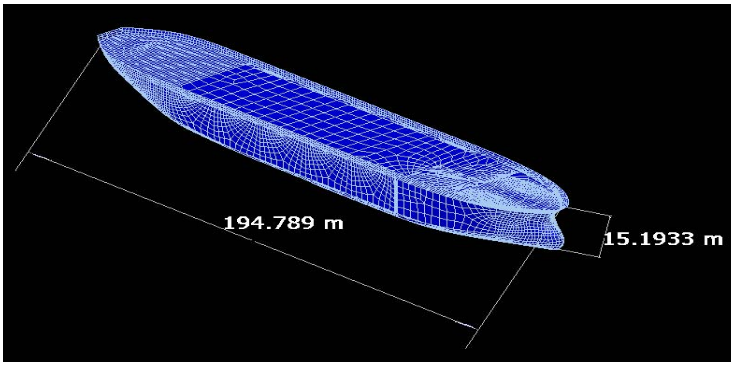


or from the toolbar button:





The select point mode must also be activated by the above button (see section Selection).



3.4 Interactive Modelling

HydroD has automated generation of pressure area elements and Morison beam element to panel correspondence (see "Element Correspondence" chapter). Bilge keels (used in roll damping calculations) can be defined by drawing on the panel model in the 3D window.

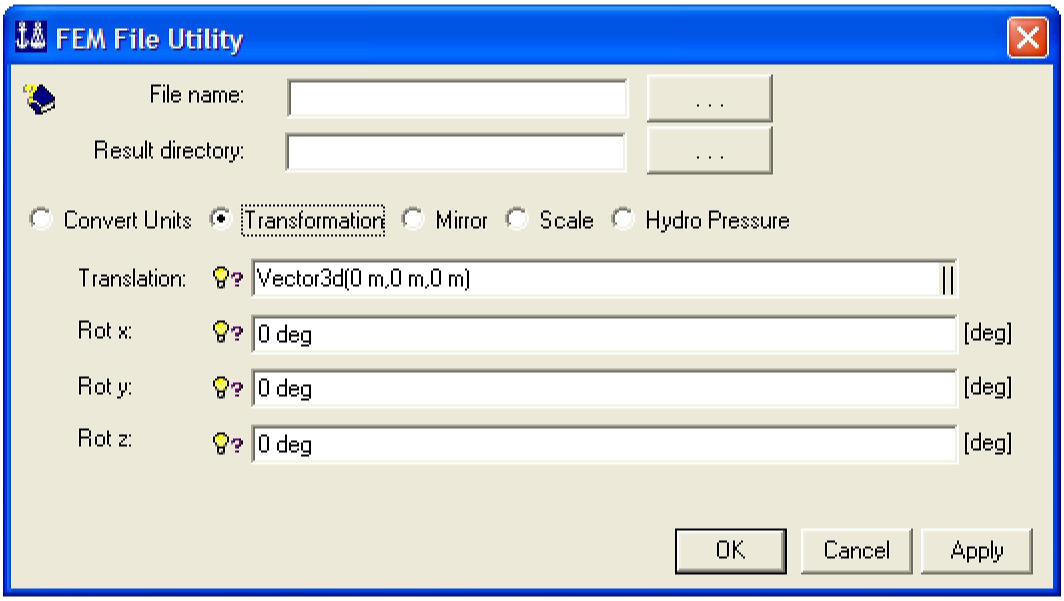
A section model can be created from scratch or modified in HydroD. Curves can be created by clicking on nodes in an imported FEM model. Section curves can also be created by automatic cutting of such a model.

3.5 Transformation of Models

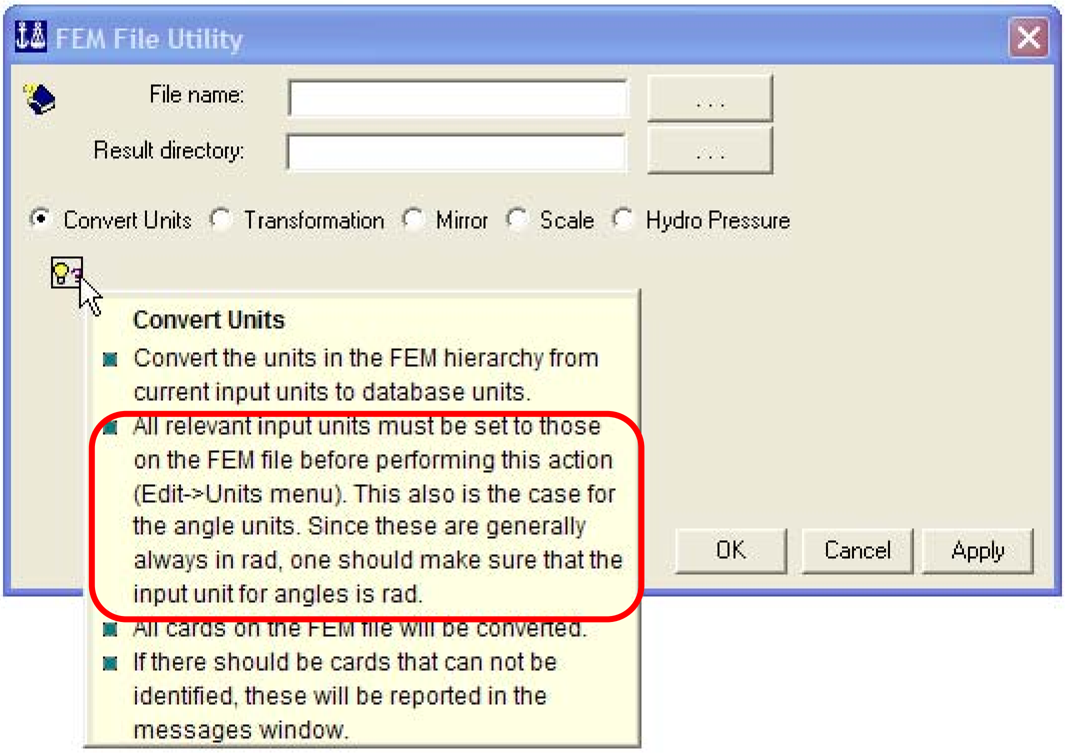
It is possible to transform a model. The transformation is performed on a Sesam Interface file (T#.fem).

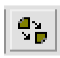
- Transformation may include both a translation and rotations
- Mirror the model about the XZ or YZ plane, or both planes
- Scale the panel model to convert it to other units (not including beam properties, shell thickness etc.)
- Hydro pressure may be added to the model. This needs a thorough verification.

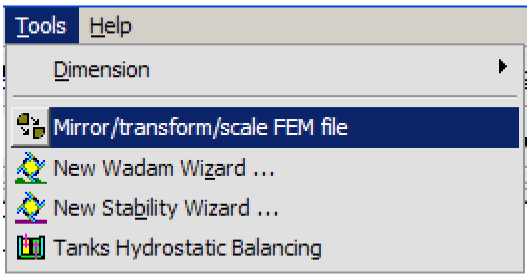
More information is found from the tool tips in the menus for the different options.



A separate type of transformation which is also available is to transform a Sesam Interface file from one unit system to another. This is done by the separate option Convert Units. The file is converted **from the Input units to the database units.**



The menu is activated from the toolbar button , or from the Tools menu:

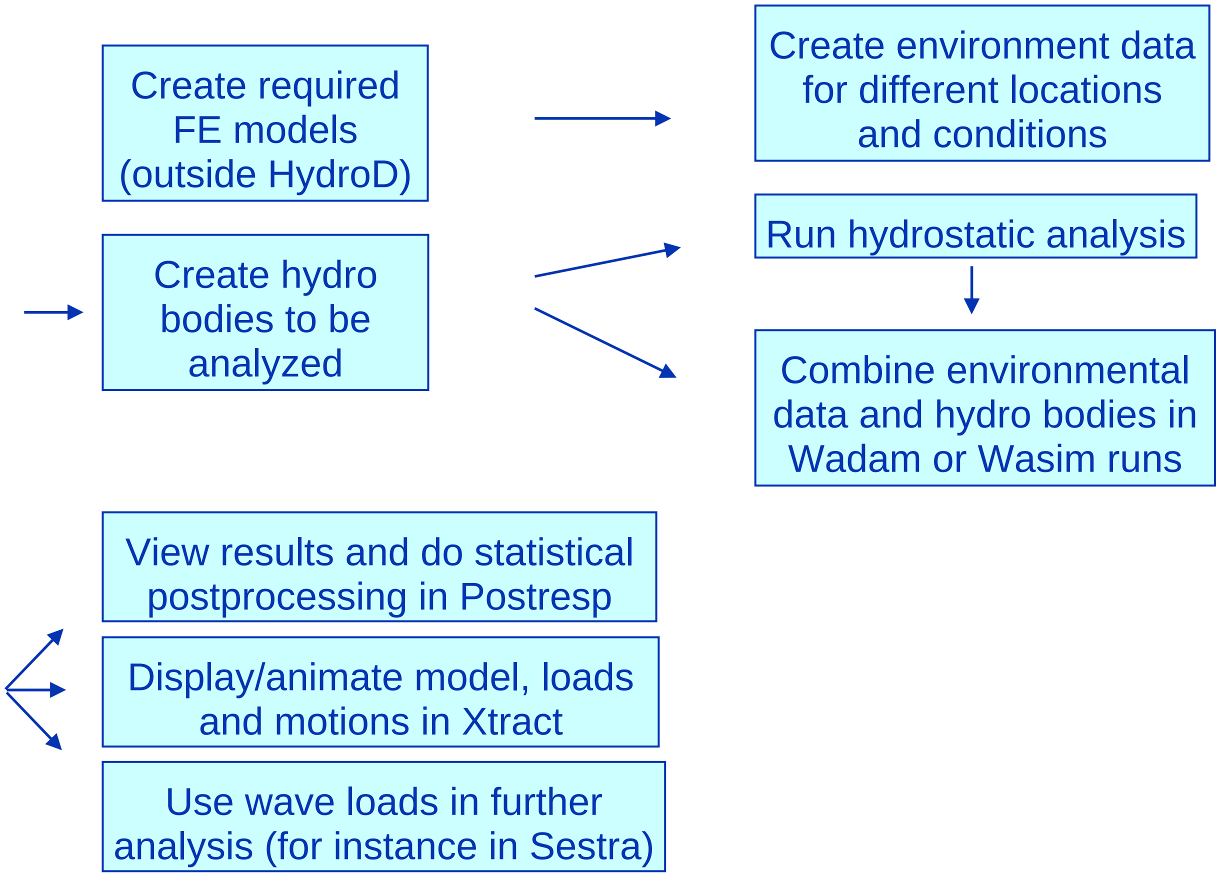


3.6 Data Checking

All input given to HydroD goes through several levels of data checking. The first level is when the data is defined. Errors that can be detected without looking at other data types will be reported. The second level of data checking is when you try to start a stability analysis or a Wadam run. All data included in that run will go through a consistency check. The third level of data checking is performed by Wadam or Wasim.

4 USER'S GUIDE TO HYDROD

4.1 The Modelling Loop



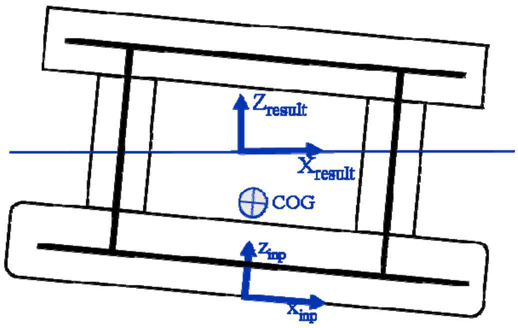
The same HydroD workspace may contain a number of environments (from different parts of the world) and several hydro bodies. When you want to execute a new analysis, it is usually best to reuse the same workspace. In this way you can reuse the environmental data and copy old hydro models or runs to reuse some of the same settings.

4.2 Coordinate Systems

HydroD uses two different coordinate systems:

- The input coordinate system. This is a right handed Cartesian coordinate system where the z-axis points upwards. All models and input data refer to this same coordinate system. Results that refer to coordinates on the body (like centre of gravity, centre of buoyancy, centre of free surface etc) refer to the input coordinate system.
- The global coordinate system. When defining a loading condition a trim, heel and change in draft is specified. The model is transformed by first performing the rotations (Euler angles with the order RY-RX, trim-heel) and subsequently changing the draft. The resulting position of the model is referred to as the global coordinate system where the still waterline is the plane at z=0.

Response results coming out from Wadam and cross sectional forces from the hydrostatics computations refer to the global coordinate system.



As described later, the mass description can be defined in either of the two coordinate systems (see section Mass Model).

When using a pre-defined model as a free surface model for off-body points, this must be modelled in the global coordinate system, i.e. z=0.

4.3 Finite Element Models

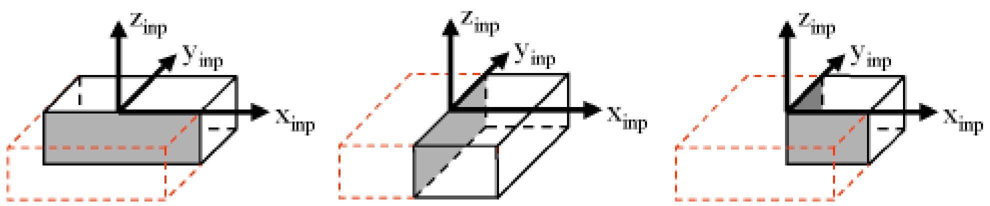
Six different types of finite element models (T*.FEM files) may potentially need to be built prior to running HydroD. Applications like GeniE and Patran-Pre may be used for the purpose. The different finite element models are:

4.3.1 Panel FEM

This FEM is used for stability calculations and for calculation of the 3D wave potential in Wadam. All panels on the wet surface must have the hydro pressure or Wet surface property load as load case number 1, pointing from the fluid onto the model. Higher order superelements may be used as panel models.

Panels are automatically cut in the waterline. This means that the user need not model exactly up to each waterline, but should rather only model one panel FEM that goes above all relevant waterlines. Similarly, if the panel model extends below the sea floor, it will be cut against the sea floor.

If the panel model is symmetric about the xz- and/or yz-plane, it is possible to model only one half/one fourth of the panel model. This will reduce calculation time and memory requirements. The model must then be defined on the positive side of the symmetry plane(s) (positive x- and/or y-coordinate):



Note that a Panel FEM must have xz-symmetry (only one half modelled) if viscous roll damping shall be included.

Wamit (.GDF) files may also be used as panel models.

If hydrostatics and stability computations shall be performed, the entire outer surface of the panel model (including any water tight super structure) should have wet surfaces/hydro pressure defined in load case number 1.

4.3.2 Morison FEM

The Morison FEM is used to calculate wave forces, added mass and drag damping using Morison theory. All elements should be modelled as 2 node beams. The Morison FEM should always be modelled in all four quadrants (i.e. no automatic mirroring). Only a single first level superelement is permitted as Morison model.

If a Morison FEM model is part of a dual model (Wadam) where the panel model has one or more symmetry planes, the Morison model should be modelled with symmetry about the relevant symmetry planes. A double symmetric panel model would for instance require that the Morison model should have symmetric elements about the xz- and the yz-plane. In the case of a dual model, one should model such that panels in the panel model can be mapped to beam elements in the Morison model.

Care should be taken on how the sections are defined on the Morison FEM. Elements that shall have different Morison properties must also have different sections (i.e. the Morison properties are connected to the sections defined on the Morison FEM).

In the case of a composite model configuration with load transfer, Morison loads are transferred to the Morison FEM.

The Morison FEM may also be used as the mass model in the analysis.

4.3.3 Mass FEM

For a floating body, one way to specify the mass is to use a mass FEM. This may contain all kinds of elements and may be a higher order superelement. The mass FEM may be the same as the structure FEM.

One should note that the mass model need not include the mass of the fluid inside the compartments. This is rather computed by HydroD and added to the mass model.

The mass model may also be defined from a Wasim mass file, .mas or .mass.

4.3.4 Structure FEM

Hydrostatic and hydrodynamic loads may be transferred to a structure FEM. The shell/solid elements of the structural model that shall receive hydro load must have the hydro pressure or Wet surface property load as load case number 1. Fluid pressures are automatically mapped from panels in the panel model to shell/solid elements in the structure FEM (i.e. the structural FEM may have a different mesh than the panel model). Higher order superelements may be used as structure models.

Compartments are employed in hydrostatic and stability computations and they can receive hydrodynamic fluid pressure from a Wadam run.

Compartments must be modelled as part of the structure FEM. The compartment must be defined as a closed volume, which means that all elements inside the compartments must have the hydro pressure/wet surface definition starting with load case number 2 for the first compartment, number 3 for the second etc. The hydro pressure/wet surface definition should point from the compartment fluid and onto the compartment walls.

In a dual and composite model configuration the structure FEM must contain all the same elements (identical with respect to element/node numbers and position in space) as the Morison FEM, but may also contain additional elements. The superelement number must also be the same, thus the Morison model must be one of the superelements in the structure (FEM) model.

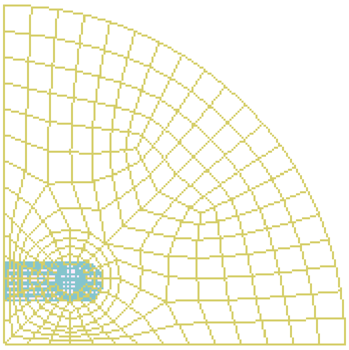
Alternatively, a single superelement composite model may be used, where the model is a complete first level superelement model, containing both the Morison parts and shell elements. In such a case, the Morison model and the structural model must be identified by the same superelement.

4.3.5 Second Order Free Surface FEM

A second order free surface FEM is required when you want to calculate second order sum and difference frequency results. (Springing is for instance typically a sum frequency phenomenon). The second order free surface FEM should be meshed with 4 node shell elements and have the hydro pressure load case number 1 pointing in negative z-direction. Higher order super elements may be used as second order free surface models.

The radius of the free surface FEM should normally be at least as large as the water depth in shallow water and the longest wave length in deep water.

The external boundary of a free surface FEM must describe a perfect circle. For a double symmetric panel model, the second order free surface mesh may look like:



Note: Second order global forces are output on the Wadam listing file. Second order pressure on body, pressure in fluid and second order wave elevation are all however only output in Wamit format.

4.3.6 Offbody Points FEM

To do free surface animation (in Xtract or other applications) an offbody points FEM can be given as input to define the free surface. This must be modelled with 4 node shell elements, located in the free surface, Z=0.0.
(This model is not the same as for the second order computation in the previous section.)

Note that HydroD is also capable of producing this file, so users do not have to create this FEM (see Offbody points paragraph).

4.3.7 Legal Elements Types

The following elements types can be used in the different FEM input to HydroD (table taken from Wadam manual).

Type of element	Number of nodes	Panel model	Morison model	Mass model	Structural model
Beam elements					
Beam	2		√	√	√
Curved beam	3			√	*
Shell elements					
Triangular flat thin shell	3	√		√	√
Quadrilateral flat thin shell	4	√		√	√
Sub parametric curved triangular thick shell	6	√		√	√

Type of element	Number of nodes	Panel model	Morison model	Mass model	Structural model
Sub parametric curved quadrilateral thick shell	8	√		√	√
Multilayered curved triangular shell	6	√		√	√
Multilayered curved quadrilateral shell	8	√		√	√
Solid elements					
Triangular prism	6	√		√	√
Linear hexahedron	8	√		√	√
Tetrahedron	4	√		√	√
Isoparametric triangular prism	15	√		√	√
Isoparametric hexahedron	20	√		√	√
Isoparametric tetrahedron	10	√		√	√
Mass elements					
One node mass element	1			√	√

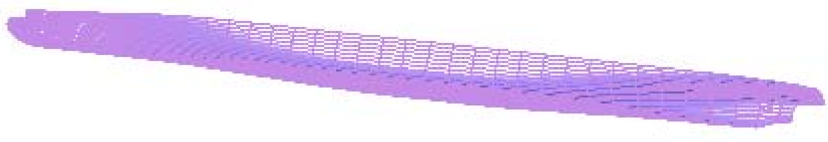
* The curved beam element may be included in a structural model, but it will not receive any direct loads (it may not be part of a Morison model).

4.4 Model Configurations

When setting up an analysis for Wadam, there are five different model configurations that may be handled in HydroD, namely panel models, Morison models, composite models, dual models and multi-body models. Before running HydroD, the user must find out what kind of a model configuration to use, and what types of finite element models (T*.FEM) this will require.

4.4.1 Panel Model Configuration

Your model contains large submerged volumes on which you want to calculate hydrostatic and hydrodynamic forces from potential theory. This is the only available option for Wasim.

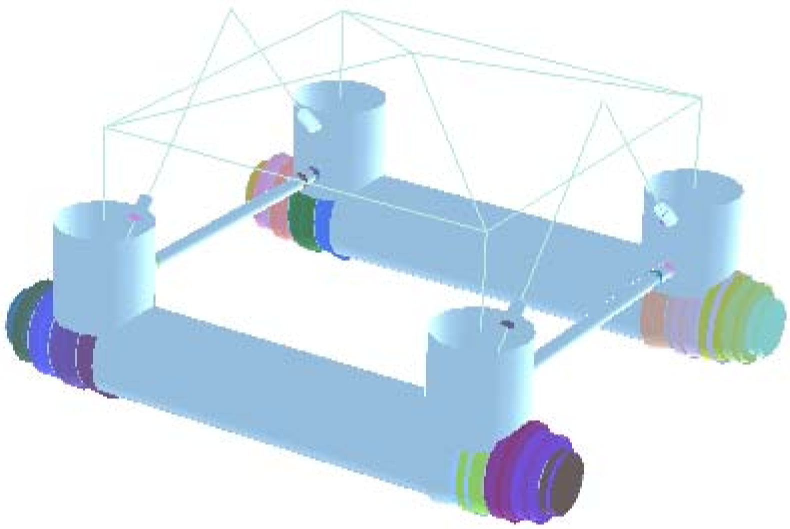


Prerequisites:

- A panel FEM or a Section Model. For Wasim only Section Model is allowed
- If loads are to be transferred, or if compartments are used, a structure FEM
- For second order results you need a second order free surface FEM

4.4.2 Morison Model Configuration

Your model consists of beam elements on which you want to apply Morison's equation (only) to calculate hydrodynamic and hydrostatic forces and motion response.



Prerequisites:

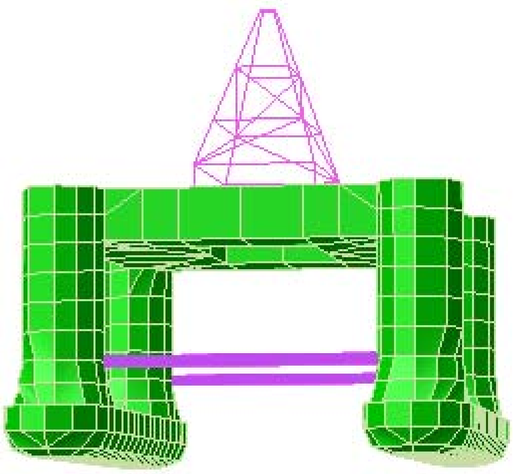
- A Morison FEM
- If loads are to be transferred to a structural model, the structural model must contain the Morison model as a superelement.

.

4.4.3 Composite Model Configuration

You want to apply both Morison equation and 3D potential theory to your structure.

Two different scenarios are very common for use of a composite model. The panel model part and the Morison model part are either completely separated from each other, or the Morison model intersects with the panel model, but the Morison elements are modelled with a "manipulated" Morison cross section. In this cross section the diameter and added mass coefficients are very small and the drag coefficients are very large.



Prerequisites:

- A Morison FEM
- A panel FEM
- If loads are to be transferred, pressures on the panel model will be applied to a structural FEM and beam loads to the Morison FEM. For further structural analysis the Morison FEM and structural FEM can be connected in a super element hierarchy.
  Alternatively, a single superelement composite model may be used, where the model is a complete first level superelement model, containing both the Morison parts and shell elements. In such a case, the Morison model and the structural model must be identified by the same superelement.
- For second order results you need a second order free surface FEM.

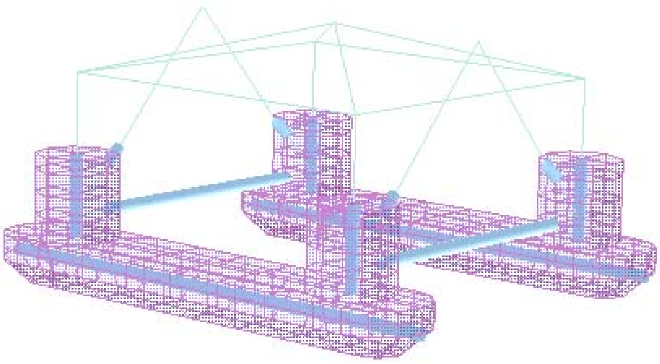
4.4.4 Dual Model Configuration

You want to apply both Morison and potential theory to your structure.

The Morison model intersects (covers the same region) the panel model, and you want all load (if any) to be transferred to a beam structural model.

Note that all wetted panels in the panel model must be connected to beams in the Morison model (meaning that the Morison model must span the entire panel model).

Also note that in a load transfer analysis, the hydrostatic load will be calculated from the Morison model alone (meaning that the Morison model must closely resemble the panel model).



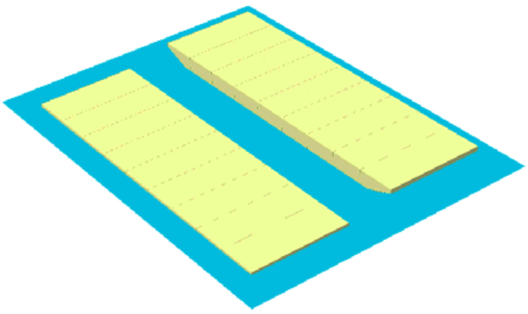
Prerequisites:

- A Morison FEM
- A panel FEM
- If loads are to be transferred, you need a structural FEM. This must contain the Morison FEM model as a superelement. Also panel pressures will be transferred to the beam structural model.
- For second order results you need a second order free surface FEM.

4.4.5 Multi-body Configuration

You want to analyze the hydrodynamic and mechanical interaction between a number of structures (bodies).

The bodies are either fixed or floating, not a combination. Stiffness and damping matrices may be defined between them.



The multi-body model is defined by selecting existing hydro models in the HydroD workspace. The hydro models in a multi-body analysis cannot contain Morison models or roll damping models.

A loading condition is selected for each body and a translation in the horizontal plane is defined. The same hydro model may thus be used many times.

A multi-body analysis cannot be defined completely from the Wadam wizard, some manual definitions are required.

Prerequisites:

- At least one Hydro model with a panel FEM
- One or more loading conditions
- A maximum number of 15 bodies may be defined

4.5 Output Files

4.5.1 Hydrodynamic Results Interface File

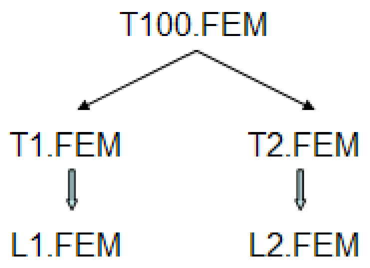
Global response results (including offbody points, panel pressures and cross sectional loads) are reported on the hydrodynamic results interface file. The file is typically used by Postresp, DeepC and Xtract. It may have one of 3 formats (as defined in the run):

- G1.SIF for formatted (ASCII) sequential file (default)
- G1.SIU for unformatted (binary) sequential file
- G1.SIN for Norsam direct access format

Wasim will only create the file in SIF format.

4.5.2 Loads Interface File

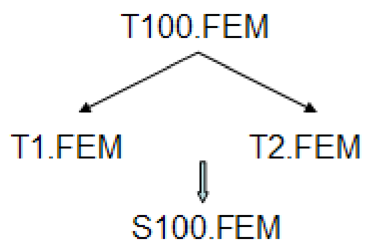
Loads transferred to the structural FEM are output on files of name L*.FEM, where * is the super element number of the first level super element on which the loads are applied. A higher order super element T100.FEM containing the two first level super elements T1.FEM and T2.FEM would for instance produce the load interface files L1.FEM and L2.FEM.



The load file can be created as a formatted file (plain text) or as an unformatted file. Please notice that for a superelement model, the loads must be assembled (combined) in Presel, in a similar manner as the models are assembled. More information is found in the user manuals for Wadam and Presel.

4.5.3 Analysis Control Data File for Structural Analysis

If a frequency domain analysis with load transfer has been performed, an analysis control data file for subsequent Sestra analysis is generated. This is named S*.FEM, where * is the superelement number of the top level superelement in the structural FEM. The file contains commands interpreted by Sestra and is mandatory for any subsequent fatigue analysis in Sesam. **Note that this file is only created when the load case numbering in Wadam is started from 1.**



4.5.4 Wadam Print File

This is named WADAM1.LIS and contains information listed by Wadam. The amount of information that shall be listed is specified in the run. Most of the results printed here are non-dimensional. The print file is usually the first thing a user inspects to see whether a Wadam analysis has been successful.

The print file has a list of contents at the beginning, which is quite useful to ease the search for the interesting results.

4.5.5 Wasim Print Files

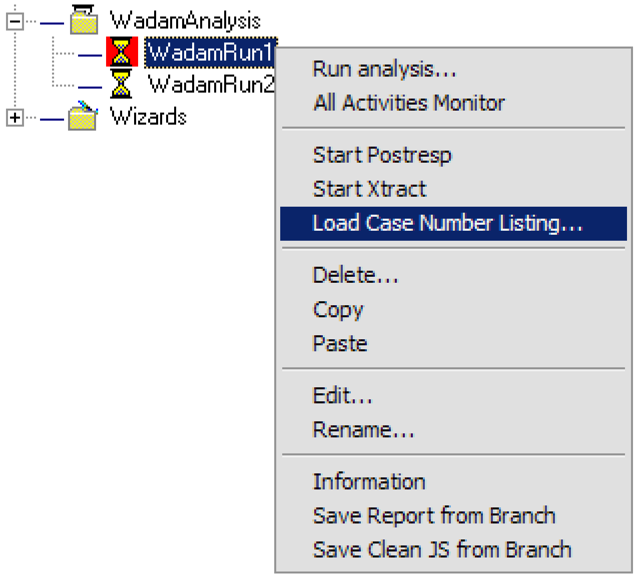
All the different modules in Wasim will also create LIS-files. In general the content of this file is quite limited. The most important information are the hydrostatic data on the Solve print file, and the matching information on the wasim_stru print file.

In addition to the LIS file the Fourier activity will create result files for rigid body motions and force RAOs (*.fmout) and sectional loads (*.ldout).

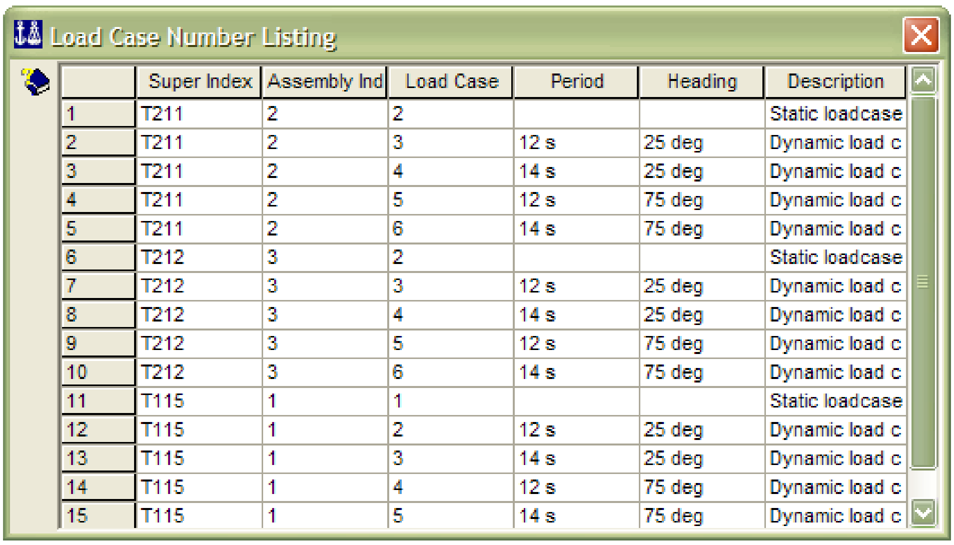
4.6 Load Case Numbering

The user is referred to the Wadam user manual for details on the numbering of the load cases produced by Wadam when transferring loads to a structural FEM.

HydroD can however list all the load cases that a run will produce. To reach the load case number listing, right click the relevant Wadam run:



The load case number listing dialog looks as follows:


Load Case Number Listing

	Super Index	Assembly Ind	Load Case	Period	Heading	Description
1	T211	2	2			Static loadcase
2	T211	2	3	12 s	25 deg	Dynamic load c
3	T211	2	4	14 s	25 deg	Dynamic load c
4	T211	2	5	12 s	75 deg	Dynamic load c
5	T211	2	6	14 s	75 deg	Dynamic load c
6	T212	3	2			Static loadcase
7	T212	3	3	12 s	25 deg	Dynamic load c
8	T212	3	4	14 s	25 deg	Dynamic load c
9	T212	3	5	12 s	75 deg	Dynamic load c
10	T212	3	6	14 s	75 deg	Dynamic load c
11	T115	1	1			Static loadcase
12	T115	1	2	12 s	25 deg	Dynamic load c
13	T115	1	3	14 s	25 deg	Dynamic load c
14	T115	1	4	12 s	75 deg	Dynamic load c
15	T115	1	5	14 s	75 deg	Dynamic load c

4.7 Program Limitations

HydroD uses dynamic allocation of memory for all objects, meaning that there should be no limitations in the number of objects/sizes of objects. However, since HydroD employs a database called Objectstore, there is a limit on the maximum total size of the database. By default this limit is 128MB, but by defining an environment variable OS_PSE_AS_SIZE, the maximum size can be increased up to approximately 600MB (OS_PSE_AS_SIZE = 600000000). If this environment variable is defined too large, the application will not start.

Also notice that the change above will make databases created prior to this change unreadable. To import such databases you must first set the variable back to the value it had when the database was created.

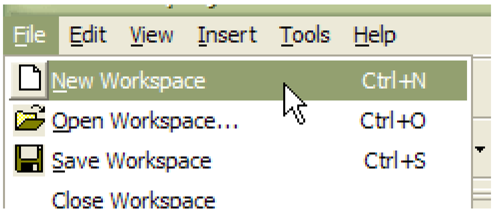
HydroD does also run Wadam and Wasim in the background, and these programs have some other limitations on the number of objects/sizes of objects. These can be found in the Wadam and Wasim user manuals.

5 DIALOG DESCRIPTION

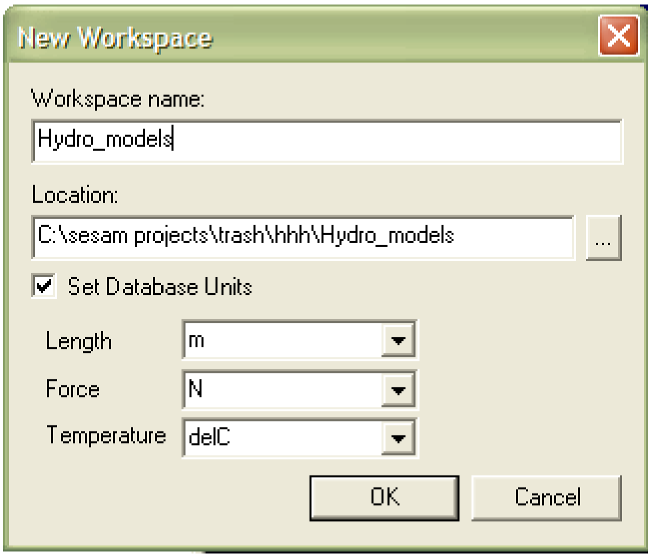
5.1 Workspace

5.1.1 New Workspace

Once the model configuration is determined and the required FEM models are built, the user may start to do the actual modelling in HydroD. The first step then is to create a new workspace. Choose "New Workspace" from the file menu (or from the toolbar):



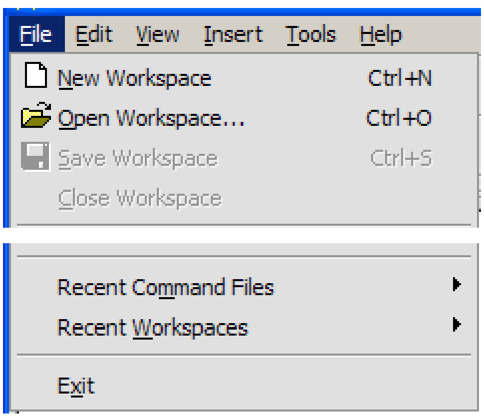
The following dialog window will pop up: (Note that this is the only time you are permitted to change the database units. Default input units may be changed at any time.)



Specify a workspace name and location and press "OK".

5.1.2 Old Workspace

An existing workspace can be re-opened from the File menu, either by use of File - Open Workspace, or File – Recent Workspaces:

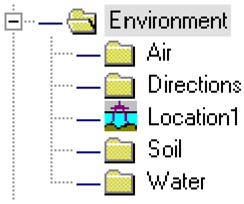


5.2 Environment Modelling

A workspace may contain environment data for a number of locations on the globe.

5.2.1 Data Organization

The environment folder contains locations, water properties and directions/direction sets:

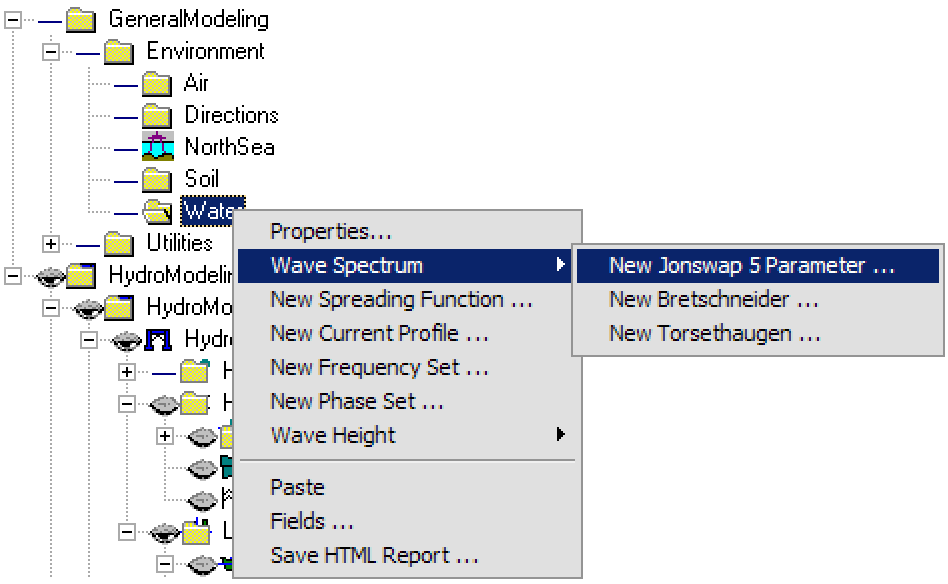


- The Air folder contains wind profiles
- The Directions folder contains directions and direction sets.
- The Water folder contains wave spectrums, spreading functions, current profiles, frequency sets, phase sets, wave height functions and regular wave sets.
- Each location (Location1, North Atlantic etc) contains a collection of site specific data/data sets.

5.2.2 Water Data Types

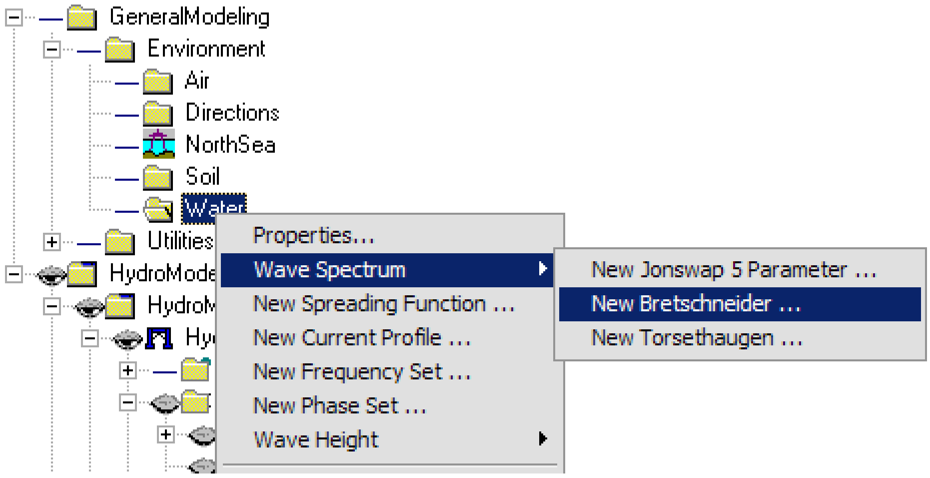
The wave spectra defined in HydroD are used for either computation of stochastic roll damping for a panel model or for linearization of drag for a Morison model. Both these features can not be used in the same Wadam run.

5.2.2.1 Jonswap Wave Spectrum



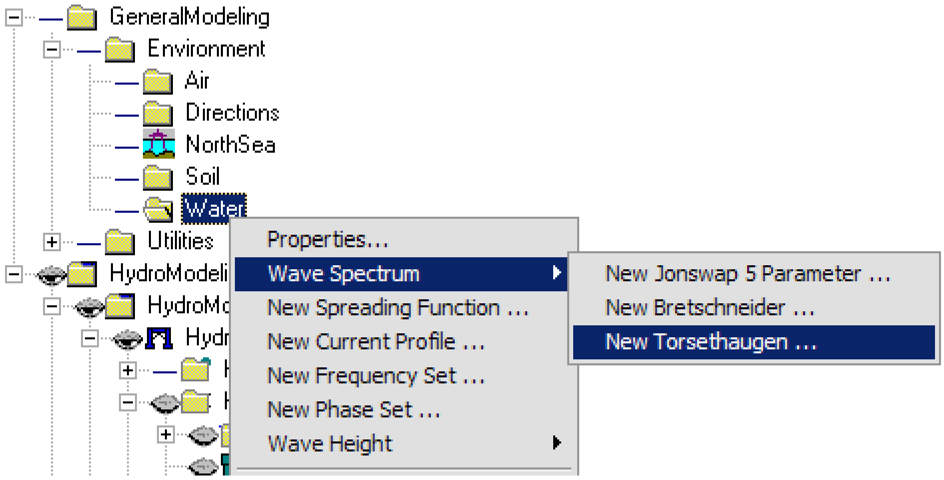
The 5 parameter Jonswap wave spectrum is defined by peakedness factor (γ), significant wave height (H_s), peak spectral wave period (T_p), left width parameter (σ_a) and right width parameter (σ_b). It is also possible to specify T_z instead of T_p.

5.2.2.2 Bretschneider (two parameter Pierson-Moskowitz) Wave Spectrums



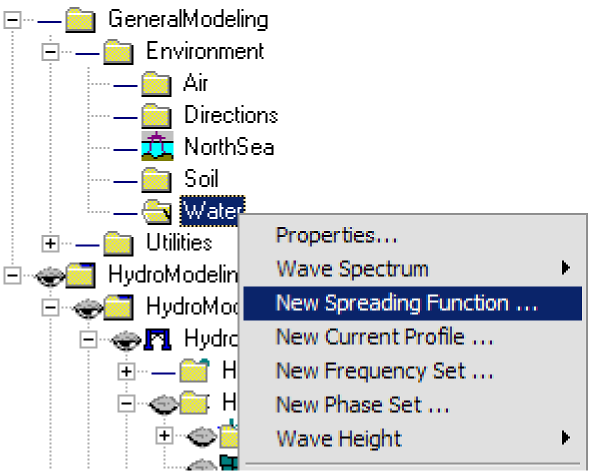
The Bretschneider wave spectrum is defined by significant wave height (H_s) and peak spectral wave period (T_p).

5.2.2.3 Torsethaugen Wave Spectrums



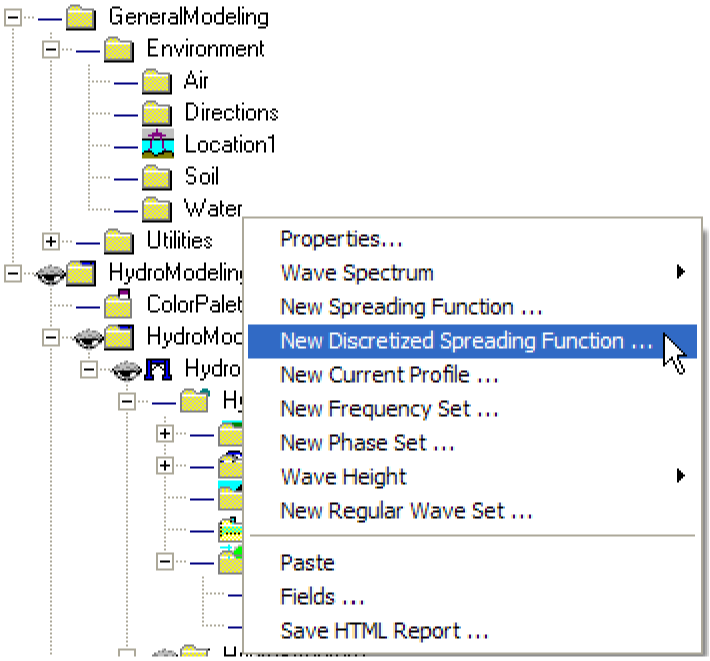
The Torsethaugen wave spectrum is defined by significant wave height (H_s) and peak spectral wave period (T_p).

5.2.2.4 Spreading Functions



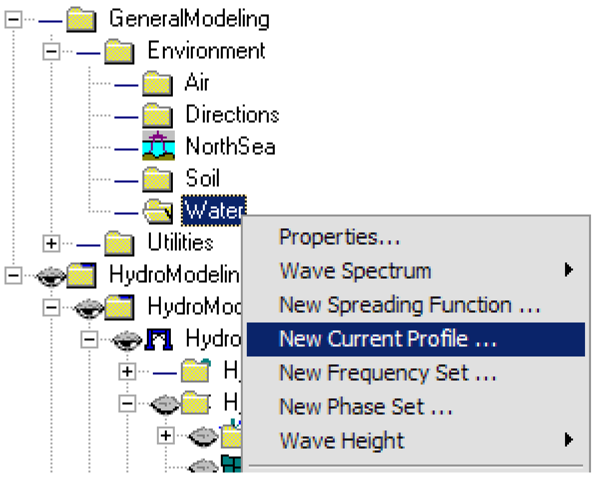
Cosine spreading functions are defined by the exponent of $\cos^n$. The wave spreading function may be used for linearization of drag for a Morison model.

5.2.2.5 Discretized Spreading Functions



Discretized spreading functions are used in the creation of Irregular Time Conditions in the case where the directions are not randomly selected.

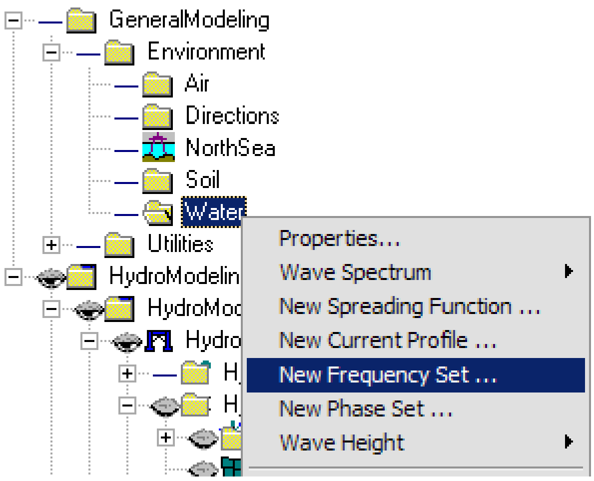
5.2.2.6 Current Profiles



Current profiles are defined by triplets of z-level, current direction and current velocity. The current direction may be given relative to wave heading or relative to the x-axis (counter clockwise).

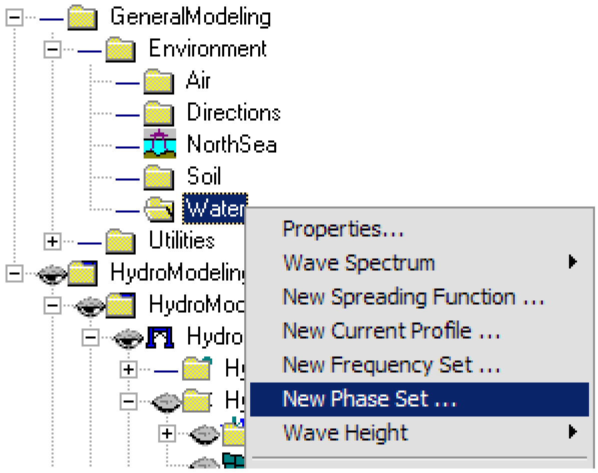
The current profile may only be used for a fixed Morison model, in a deterministic ("time-domain") analysis with Wadam.

5.2.2.7 Frequency Sets



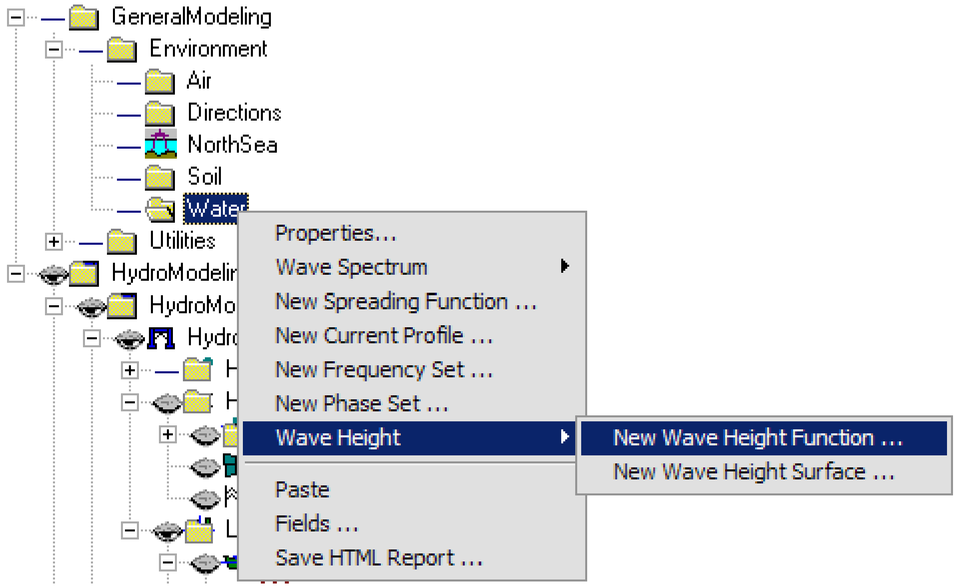
Frequency sets are a list of periods, wave lengths or wave angular frequencies. One should note that Postresp requires constantly spaced wave angular frequencies for second order statistical post processing operations.

5.2.2.8 Phase Sets



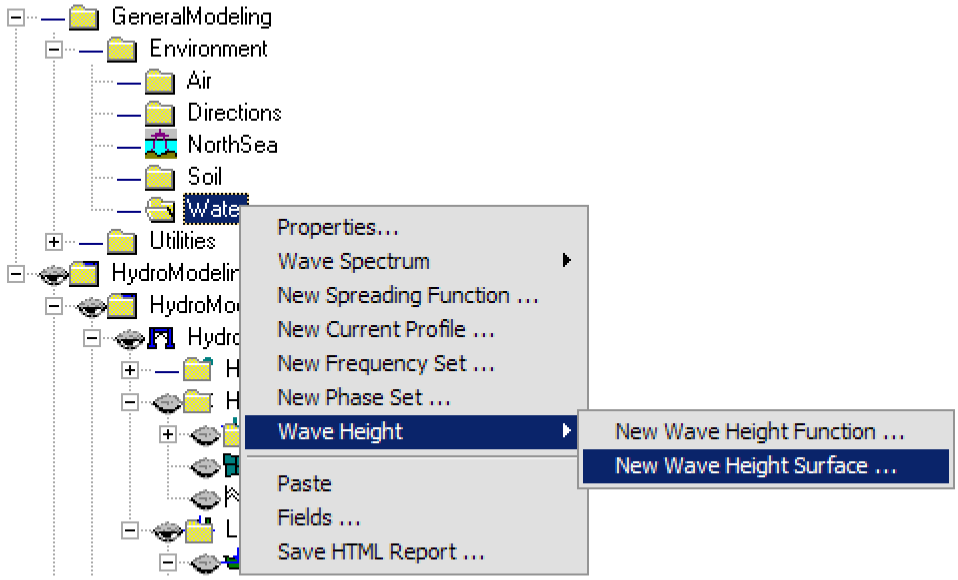
Phase sets are a list of wave phase angles at which wave loads shall be evaluated in a deterministic ("time domain") analysis with Wadam. Wadam currently enforces a limit of maximum 8 phase angles.

5.2.2.9 Wave Height Functions



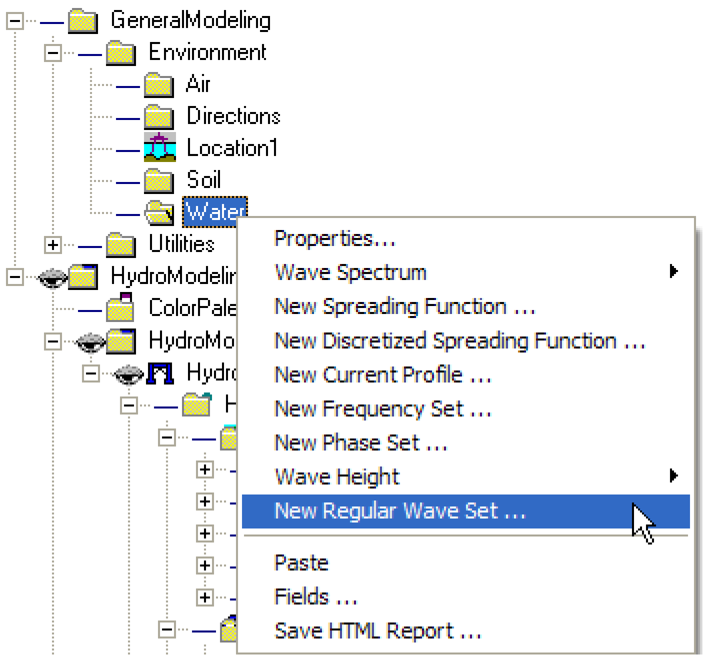
Wave height functions are defined by pairs of wave period and wave height (double amplitude). They are used to extract the wave height in a deterministic ("time domain") analysis in Wadam. Wave heights for periods not given in the wave height function are extracted by linear interpolation.

5.2.2.10 Wave Height Surfaces



Wave height surfaces give wave height as a function of both period and wave heading. They are used by Wadam in drag linearization (as an alternative to stochastic linearization by defining a wave spectrum). A regular matrix of wave heights are given as input where the periods are defined by a first period, period step and number of periods, and the headings similarly by a first heading, a heading step and number of headings. In between values of wave heights are found by linear interpolation.

5.2.2.11 Regular wave sets

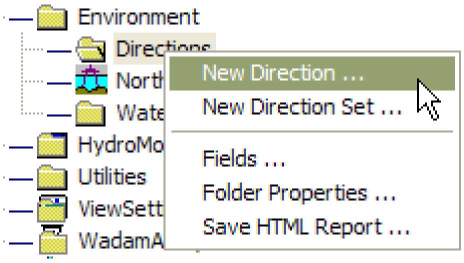


A regular wave set is a set of user defined discrete harmonic wave components. Each component is defined by amplitude, period, direction and phase angle. This is the wave input to a Wasim time domain analysis.

5.2.3 Directions Data Types

The directions data define the wave propagation directions included in the analysis and the main wave direction used for definition of wave spreading in a sea state.

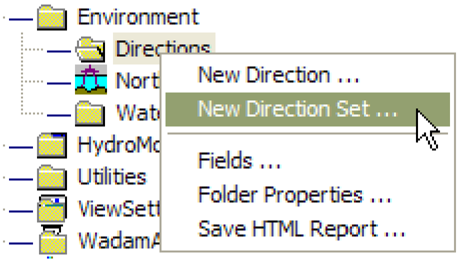
5.2.3.1 Directions



This is used for definition of the main wave direction when using wave spreading.

Directions are defined as angles counter clockwise from the x-axis. A direction of 0 degrees means that the waves propagate along the x-axis in the positive x-axis direction.

5.2.3.2 Direction Sets

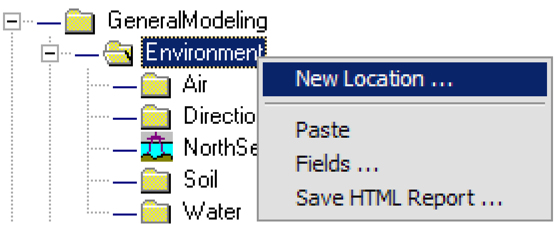


Direction sets define a list of wave propagation angles.

The angles are counter clockwise relative to the x-axis. A direction of 0 deg means that the waves propagate along the x-axis in the positive x-axis direction.

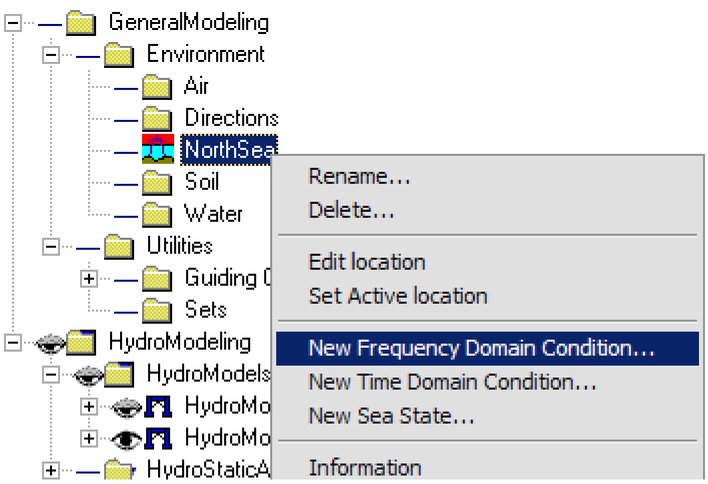
5.2.4 Location Data Types

5.2.4.1 Locations



Locations contain gravity, air & water density, air & water kinematic viscosity and water depth. Under hydro locations one may define hydro conditions and sea states.

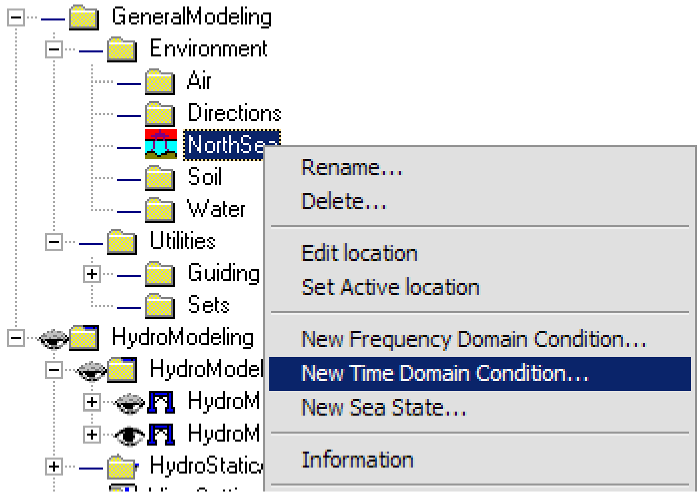
5.2.4.2 Frequency Domain Conditions



Frequency domain conditions contain the data used to discretize the frequency and heading domain in a frequency domain wave load analysis (frequency set and direction set).

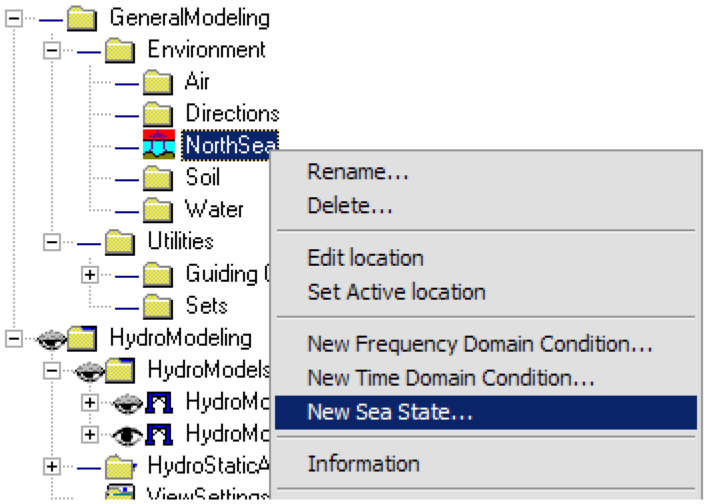
The Wave Amplitude setting is only relevant for Wasim. This amplitude will be applied to all wave components in the Wasim runs.

5.2.4.3 Time Domain Conditions



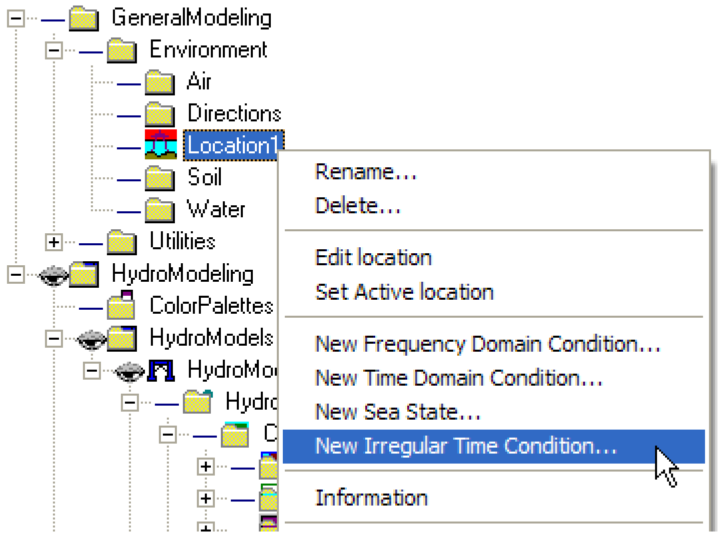
Time domain conditions contain the data specifying the regular waves and current used in deterministic ("time domain") analysis (a frequency set, a direction set, a phase angles set, a wave height function and a current profile).

5.2.4.4 Sea States



Sea states contain the data required to produce an irregular sea (wave spectrum, spreading, sea state duration etc). Sea states are used for stochastic roll damping and drag linearization calculations.

5.2.4.5 Irregular Time Conditions

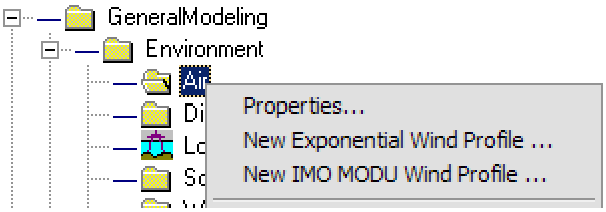


Irregular Time Conditions are somewhat equivalent to Regular Wave Sets. But instead of specifying the set of harmonic wave components directly the user specifies a Sea State and defines some setting for automatically creating a regular wave set which is a realization of the selected sea state. When a Wasim time domain analysis is specified and an irregular time condition is selected this realization will be prescribed as wave input to the Wasim analysis. A sea state can be created as a combination of a swell and a wind driven component. Separate direction and wave spectra are specified

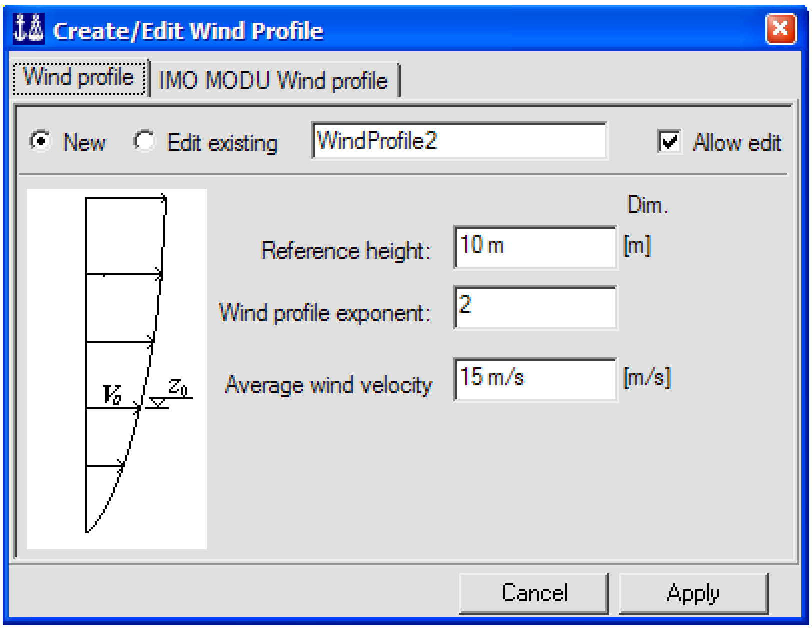
for the two components.

5.2.5 Air Data Types

The Air data define the wind profiles used in the computation of wind heeling moment curve in a stability analysis.

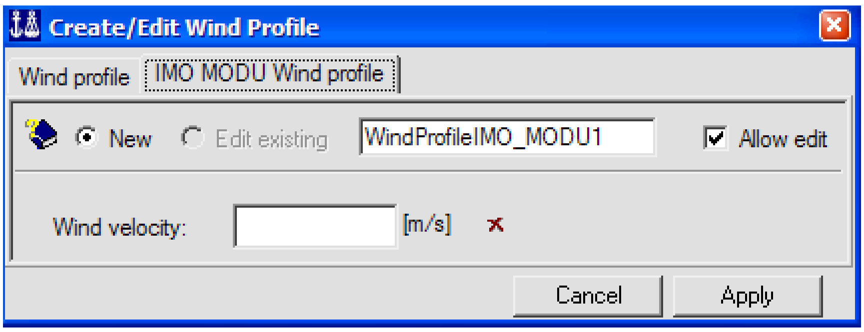


5.2.5.1 Exponential Wind Profile



The wind profile is defined from an average wind velocity at a given reference height, using a given exponent.

5.2.5.2 IMO MODU Wind Profile



The wind profile is defined as given by the IMO MODU Code. Further information is given in the tool tip of the menu.

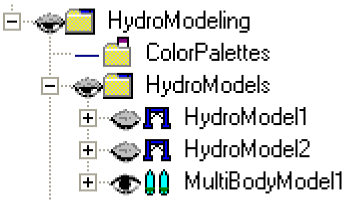
5.3 Hydro Models

A workspace may contain a number of hydro models. Currently only one hydro model may be analyzed in Wadam at a time.

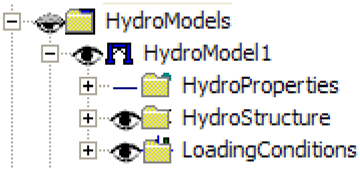
When a multi-body analysis (Wadam) is wanted, a separate multi-body model is defined from one or more hydro models.

5.3.1 Data Organization

One or more hydro models may be defined in the ‘HydroModels’ folder.

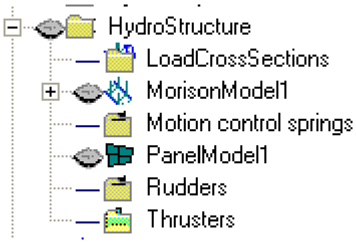


A hydro model contains 3 main sub folders:

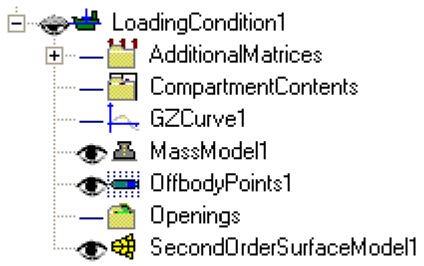


- The hydro properties folder may contain properties for compartments, Morison model, Rudders and thrusters and wind.

- The hydro structure folder contains all loading condition independent models and element types that may be defined on a hydro model. The Panel Model may be replaced by a Section Model.

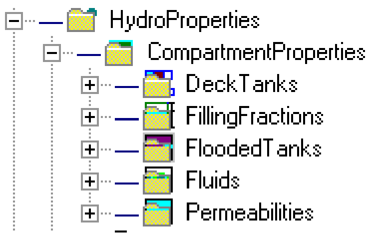


- The loading conditions folder contains all data related to different draughts of the hydro model.



The data types are described in more detail on the following pages.

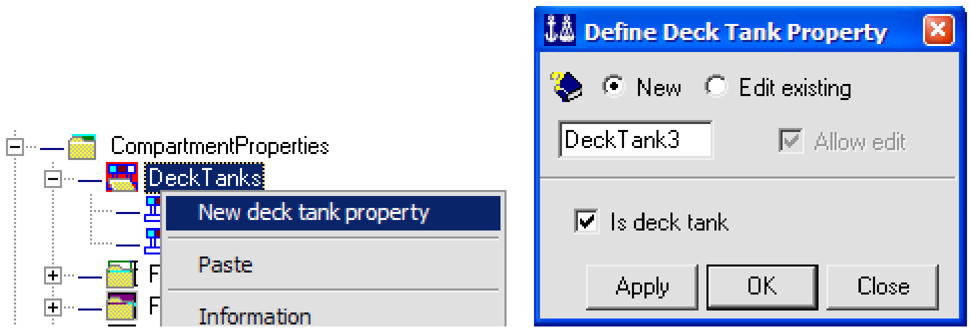
5.3.2 Compartments Properties



The Deck Tank and Permeability properties are assigned to compartments (in the Structure Model folder).

The Filling Fraction, Flooded Tanks and Fluid properties are assigned to compartment contents (in the Loading Condition folder).

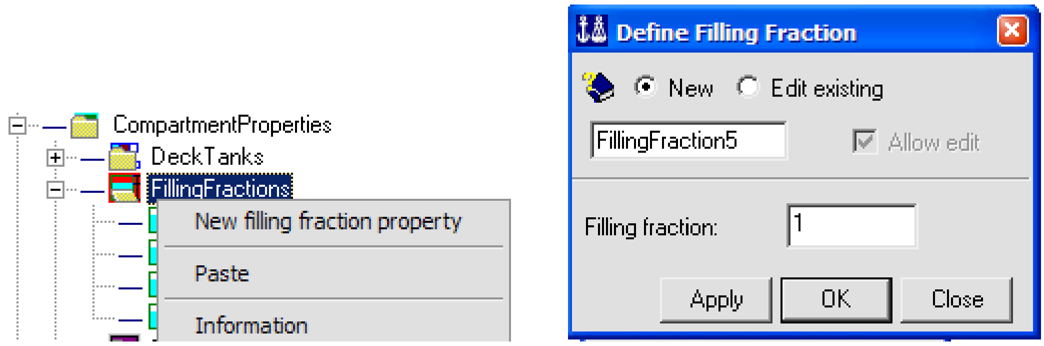
5.3.2.1 Deck Tanks



Used to specify whether a tank is part of the deck structure on a mobile offshore unit or not. The default setting is that a compartment is not a deck tank.

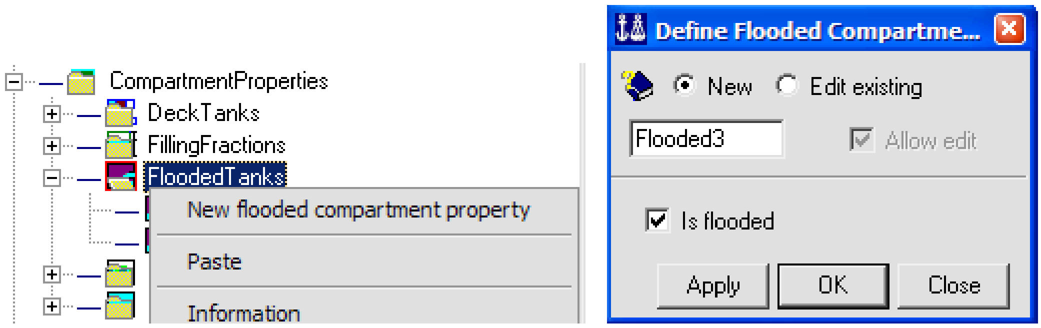
Deck tank setting is not relevant for ships. Deck tanks are only used in connection with the NMD stability code check. From the stability wizard this dialogue will thus only be started if the NMD code check has been selected.

5.3.2.2 Filling Fractions



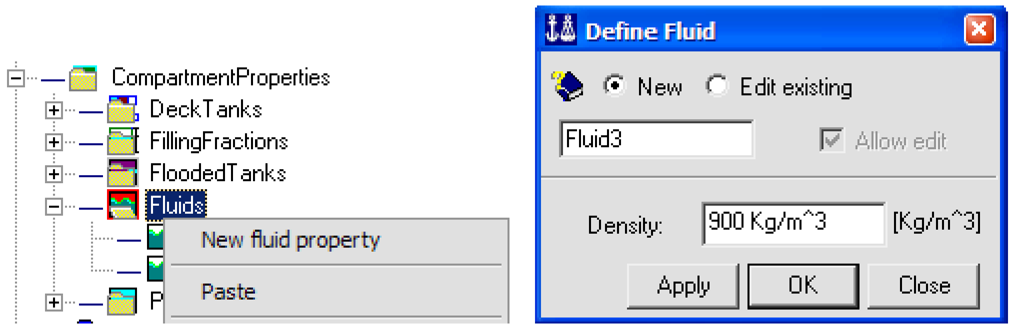
A Filling Fraction property is defined for each different filling ratio of a compartment. The Filling Fraction will be assigned to one or more compartment contents. The filling ratio is visualized in the display of the compartments.

5.3.2.3 Flooded Tanks



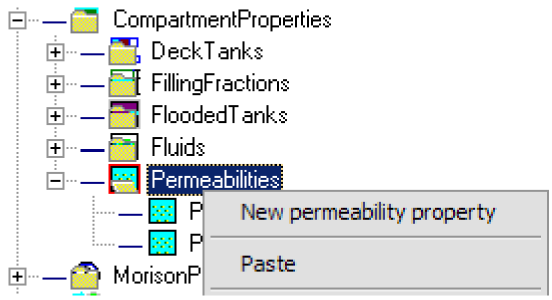
Compartments may be defined as flooded. If so, the internal free surface level will coincide with the external free surface level. The flooding will be assigned to one or more compartment contents.

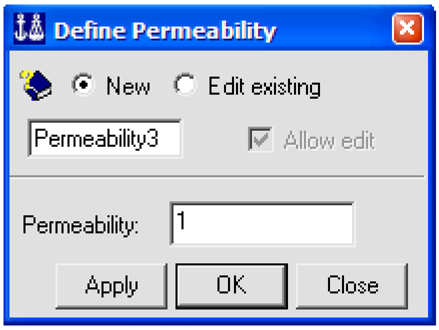
5.3.2.4 Fluids



The fluid density is given as a property to be assigned to compartment contents.

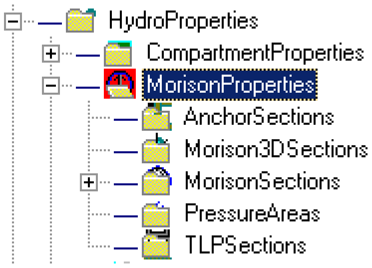
5.3.2.5 Permeabilities





The permeability describes the fraction of the compartment volume that can be filled with fluid.

5.3.3 Morison Properties



This folder contains properties of the Morison model. The details are described in the Morison Model section.

5.3.4 Rudder Properties

This folder contains properties for rudders. Rudders can be used for motion control in a Wasim analysis.

The autopilot model in Wasim is very simplified and should only be regarded as a numerical tool to control horizontal motions. Thus the rudders defined here do not necessarily have to reflect the actual physical rudder on the vessel.

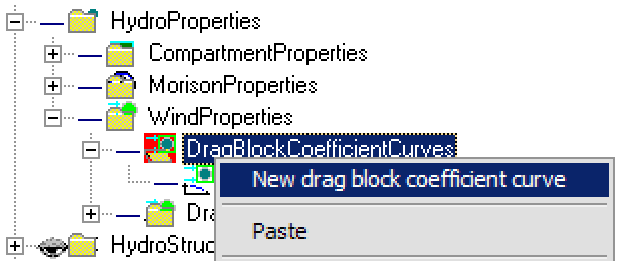
5.3.5 Thruster Forces

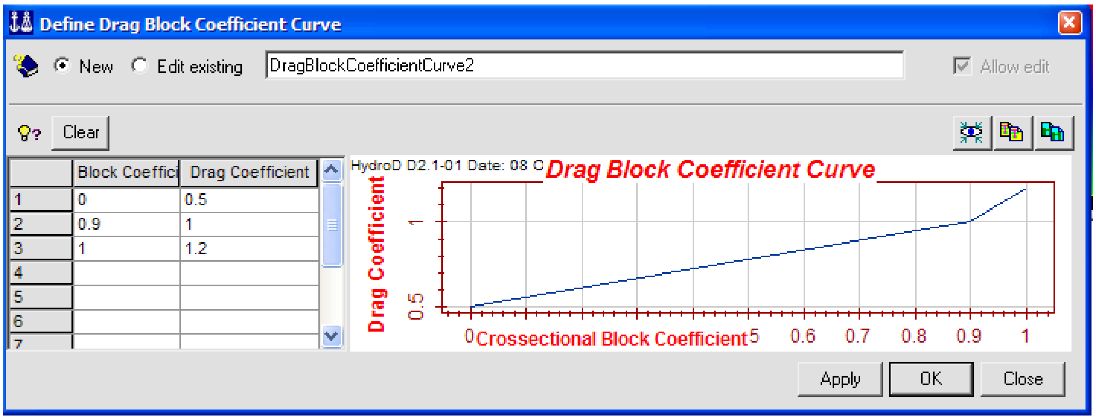
Different types of thrusters with different force characteristics can be defined. Thrusters can be used in stability analyses.

5.3.6 Wind Properties

The wind properties are used in the computation of wind heeling moment. This is based on drag coefficients and drag block coefficients.

5.3.6.1 Drag Block Coefficient Curve

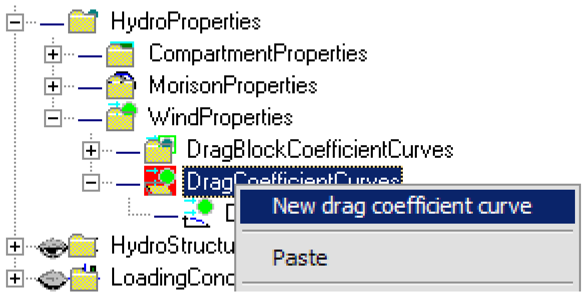


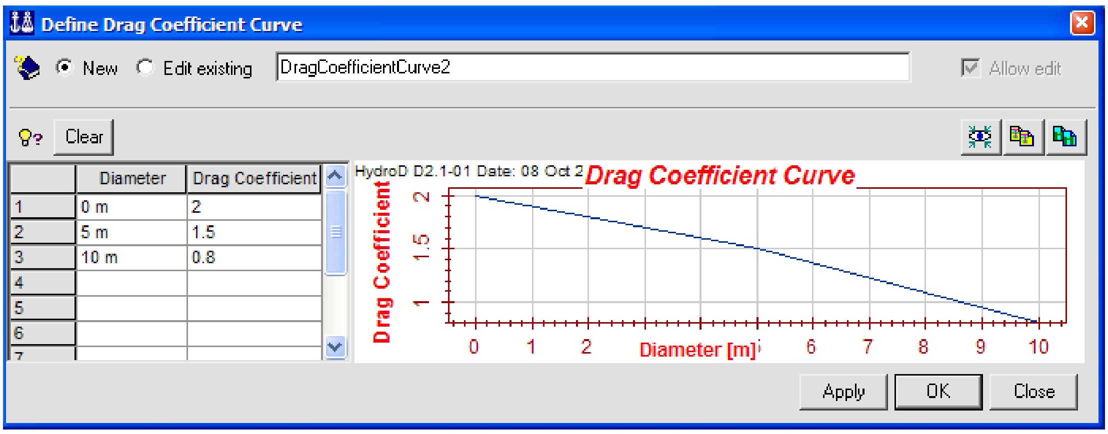


A drag coefficient is given for defined cross sectional block coefficients. Linear interpolation is used to extract values between the points in the curve.

A more detailed description of this is given by pointing at the tool tips of the dialog window.

5.3.6.2 Drag Coefficient Curve

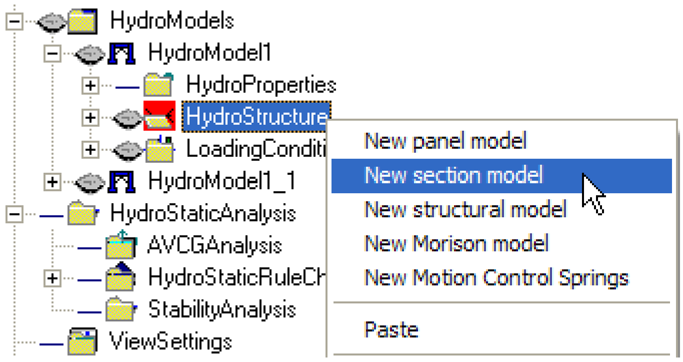




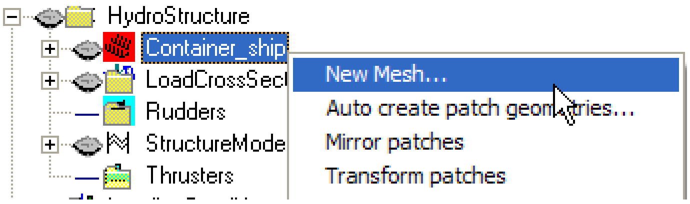
A drag coefficient is given for defined diameters. Linear interpolation is used to extract values between the points in the curve.

A more detailed description of this is given by pointing at the tool tips of the dialog window.

5.3.7 Section Model



The section model can be created from scratch or imported from a file. The file extension of this file should be .pln, hence it is often referred to as the pln-file.



Based on the Section Model HydroD can create a mesh (see additional description in the section Setup activity). This mesh can be used as a panel model the in the same way as the Panel Model object described below.

5.3.8 Import DXF geometry to section model

Dxf files containing simple polylines may be imported into the section model:

- Create a new section model from scratch.
- Right click the section model and select "Import patch...".
- Select the dxf file to import. The resulting curves might be in the wrong order, use the edit patch dialog to reorder the curves.

The DXF import tool supports simple polylines of the following DXF entity types

- LINE
- POLYLINE
- LWPOLYLINE

Extrusions and transformations from ECS to WCS coordinate systems are supported.

Note that splines/curves should be exported as polylines in the source application, and make sure that each curve is not split in multiple segments.

5.3.9 PLN file format

The PLN file format is using formatted text files, defined by the following schema

Description	Example
Name of vessel	`Container ship`
Number of patches	`7`
PatchCodes (one for each patch)	`1 1 1 1 1 1 10`
Symmetry flag	`1`
AP and FP x coordinates	`0 278`
for each patch: Patch name	`Bow upper`
Number of curves	`6`
for each curve: number of points	`6`
for each point: X Y Z	`276.25 0 9.66`
X Y Z	`275.5 0 9.84`
...	`...`

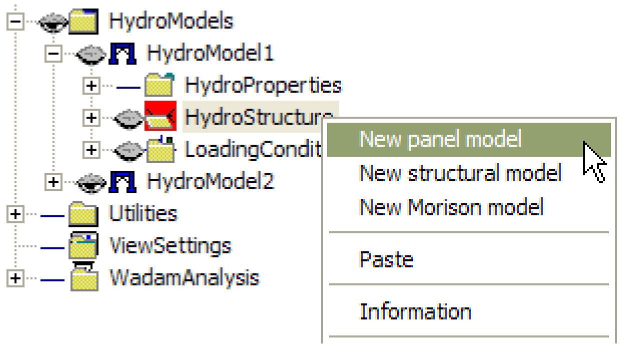
Patch codes

2	Completely dry
1	Port side wet
-1	Starboard side wet
0	Fully submerged
10	Port side wet, no waterline
-10	Starboard side wet, no waterline

Name of vessel and patch name

Use only letters from A-Z and digits 0-9

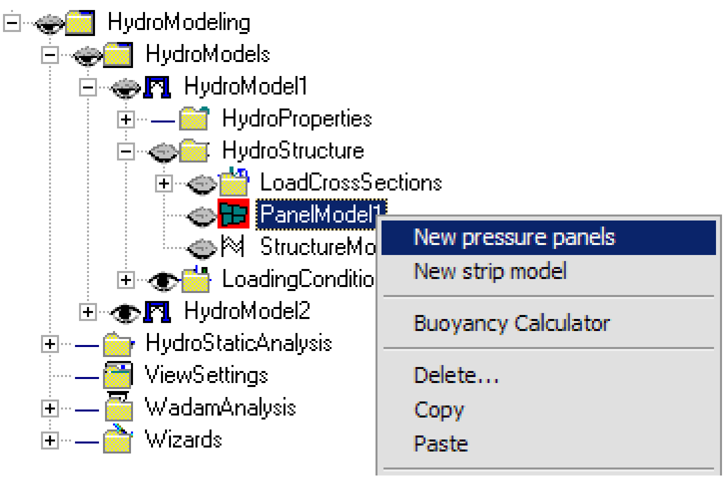
5.3.10 Panel Model

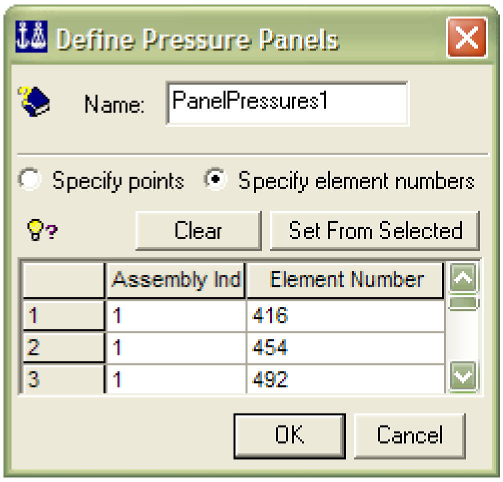


Defined by T*.FEM file or Wamit .GDF file (see paragraph about Panel FEM).

5.3.10.1 Pressure Panels

A selection of panels may be defined for statistical post processing (as opposed to load transfer) of the panel pressure. Such panels can also be defined from the mesh created from the Section model.

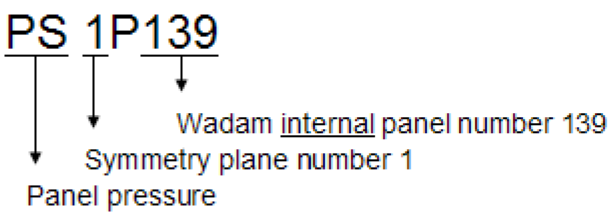




Panels may be selected in the 3D window (see “Selection” paragraph) and set into the table.

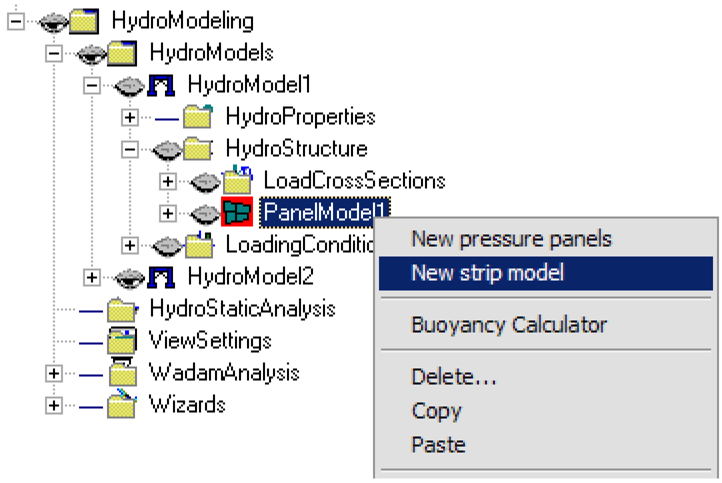
Panels may be specified by element number or centre coordinate. When element numbers are chosen, the coordinate of the panels may sometimes change if the panel model is remeshed. If a centre coordinate is chosen, the panel with its centre closest to the centre coordinate will be used, independent of its number.

Note that for statistical post processing in Postresp, the panels are named according to the internal panel number in Wadam:



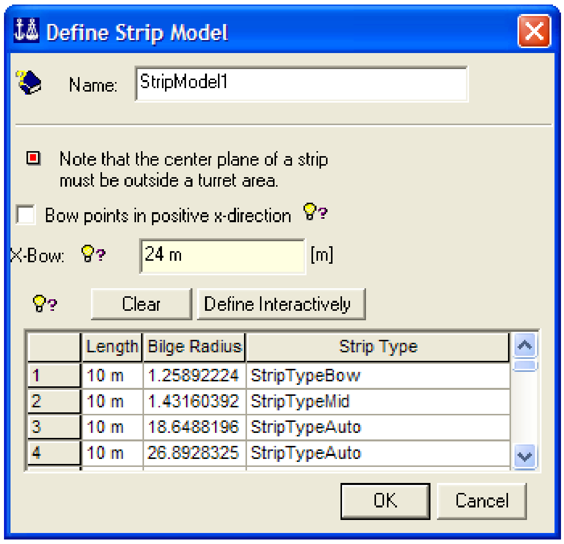
To find the external panel number that this corresponds to, the user should specify “Normal Print” (or more) in the run dialog. In the Wadam print file (see “Wadam Print File” section) the user will then get a listing of all panels including the internal panel number from Wadam and the external panel number.

5.3.10.2 Strip Model

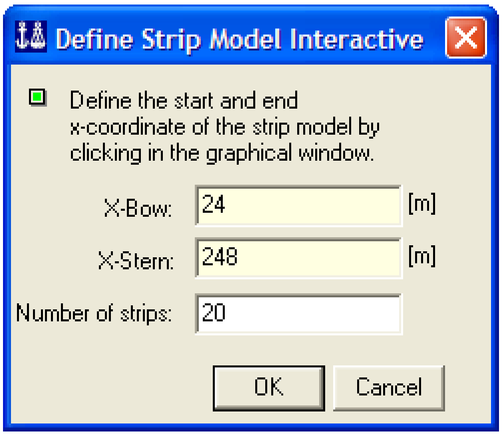


A strip model is used for roll damping calculations in Wadam. Each strip is defined by its length, the bilge radius and the type of strip. See dialog tool tips for details on strip types etc.

For some models it may be correct to specify the strip type in stead of using the automatic definition.



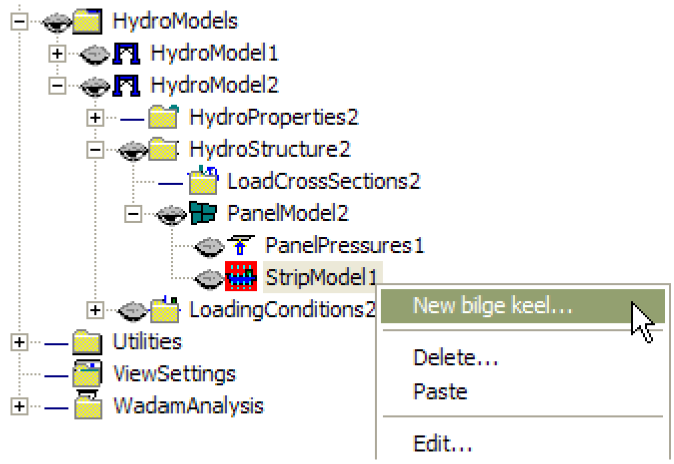
If the “Define Interactively” button is pressed a new dialog will appear:



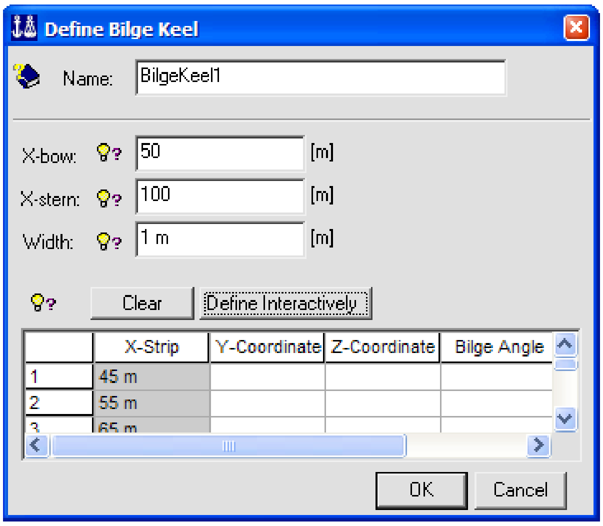
By positioning the cursor in the “X-Bow:” or “X-Stern” edit controls the user may select the start coordinate of the first strip and the end coordinate of the last strip by selecting on the model in the 3D window. When “OK” is pressed, the original strip model dialog will reappear, filled with data extracted from the model.

Notice that by setting the print option for Wadam to more than Normal print, detailed information on the strip model and bilge keel will be included.

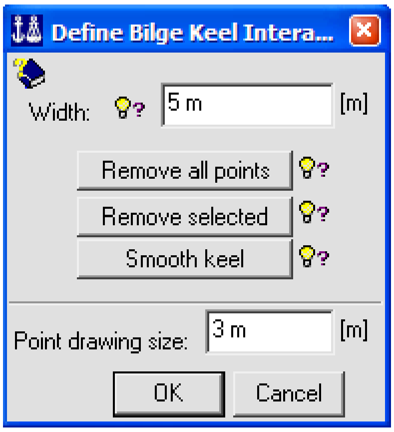
5.3.10.3 Bilge Keel



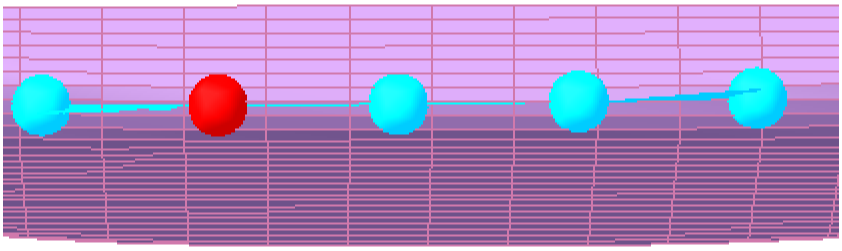
Bilge keels are used in viscous roll damping calculations. They are defined by a start and end x-coordinate and a constant width along the entire length. In addition the user has to specify y-coordinate, z-coordinate and bilge angle at the centre of each strip the bilge keel intersects (see tool tip for more information). When an x-start and end coordinate has been given, the table in the dialog is automatically filled with the correct number of rows and the x-coordinate of the centre of the strips:



If the “Define Interactively” button is pressed, a new dialog will appear:

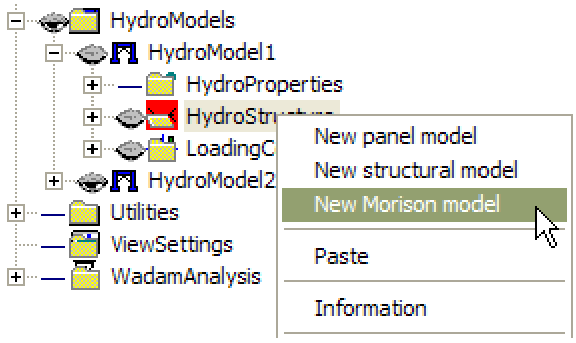


The user should now define the bilge keel by clicking (LMB) on the panel model.



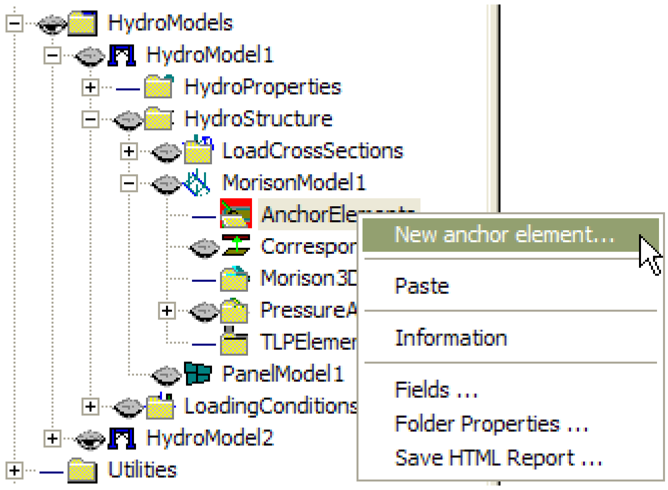
Each click will define a new point along the bilge keel. Points can be moved by pressing LMB on top of them and dragging the mouse cursor around. Multiple points can be moved simultaneously by keeping shift pressed, click on one and one point and on the last point keep LMB pressed and drag the cursor around.

5.3.11 Morison Model



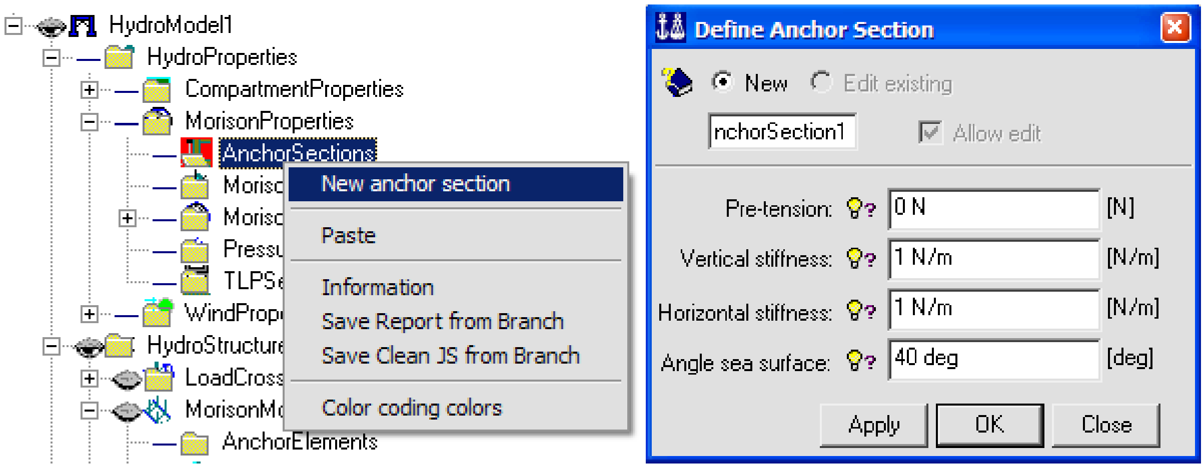
Morison models are defined by a T*.FEM file. (See paragraph about Morison FEM.)

5.3.11.1 Anchor Elements



Anchor elements with linear stiffness characteristics may be defined in HydroD. These must be connected to nodes in the Morison model. An element is defined by a fairlead node, windlass node, angle with x-axis and anchor section. The fairlead and windlass may optionally be one point.

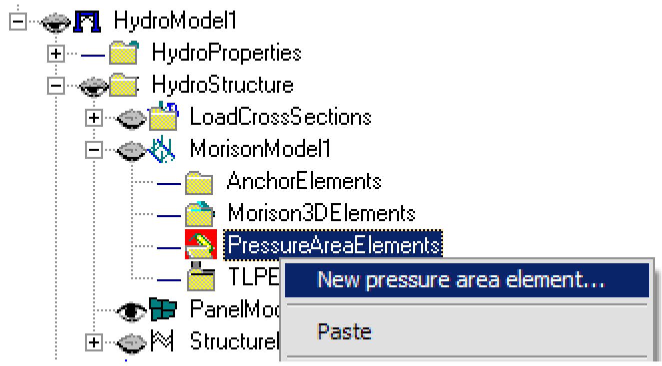
5.3.11.2 Anchor Sections



Characteristic common properties of anchor elements including:

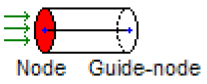
- Static mooring line force (pre tension)
- Vertical stiffness
- Horizontal stiffness
- Angle with the sea surface

5.3.11.3 Pressure Area Elements



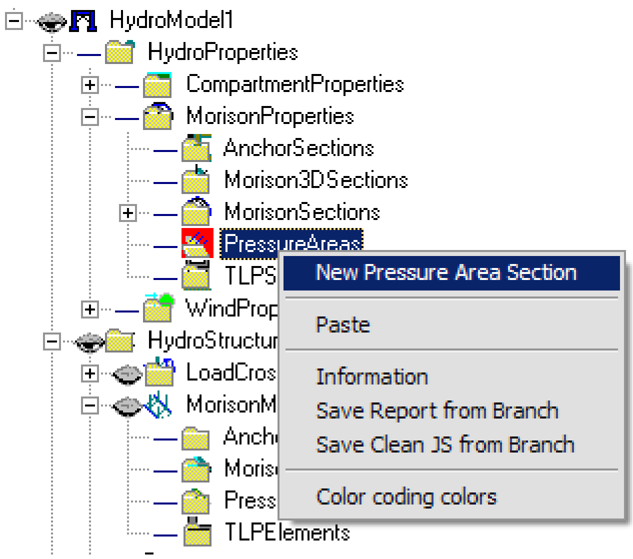
Morison beams only define hydrodynamic load transversely on the beams. Pressure area elements are put on beam ends to account for the Froude-Krylov hydrodynamic pressure acting there. On dual models pressure area elements are also used to calculate hydro-static axial load on beams.

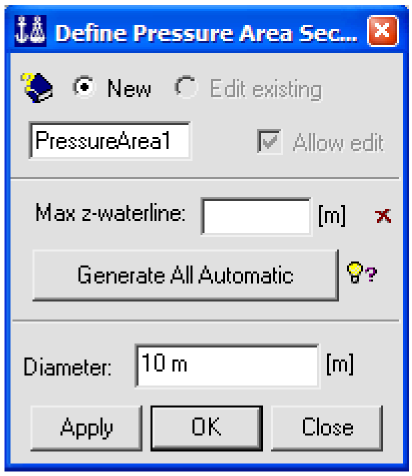
Pressure area elements are defined by a node, a guide node and a pressure area section.



All pressure area elements and sections may be defined by an automatic algorithm. Several pressure area elements may be defined in the same node.

5.3.11.4 Pressure Areas

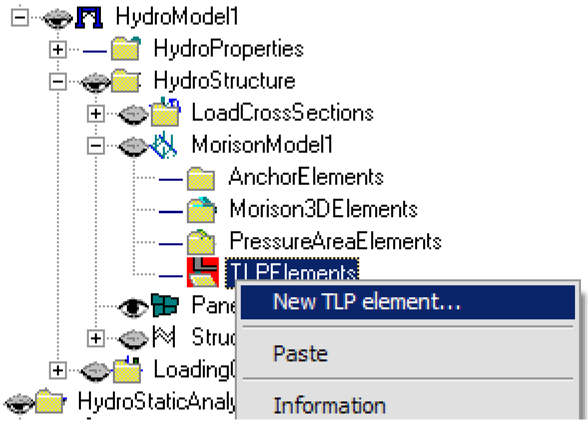




Define Area of pressure area elements. All pressure areas may be defined by an automatic algorithm.

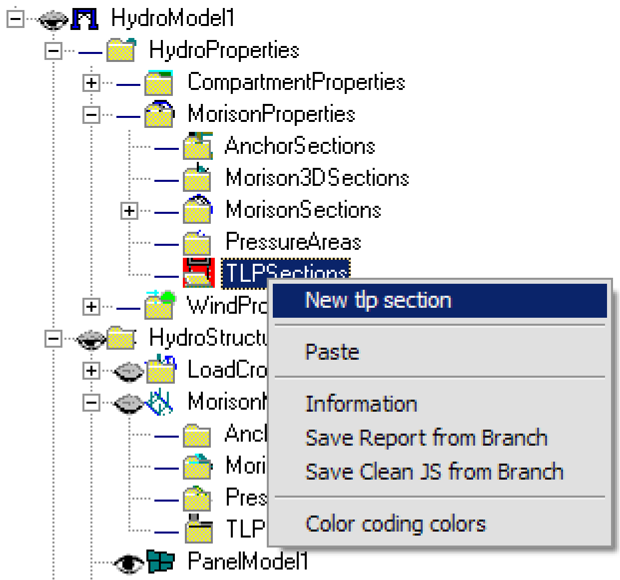
Notice that when the automatic option is used both Pressure area sections and pressure area elements are created.

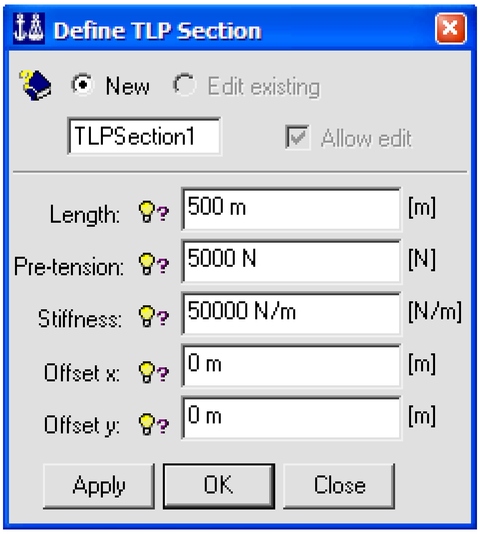
5.3.11.5 TLP Elements



Tension leg anchor elements with linear stiffness characteristics may be defined in HydroD. These are defined by a node, from which they go down, and a TLP section.

5.3.11.6 TLP Sections

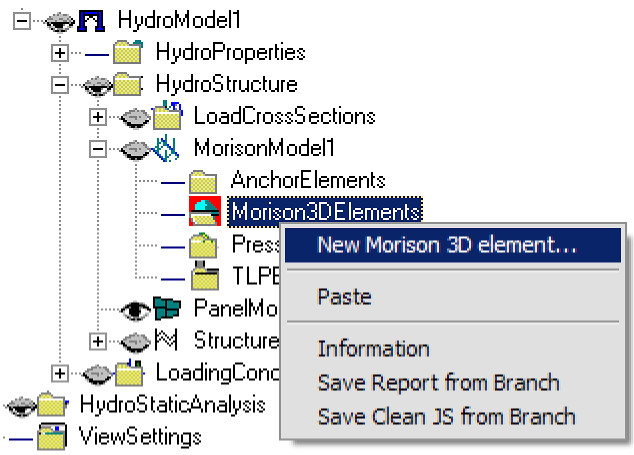




Characteristic common properties of tension leg anchor elements including:

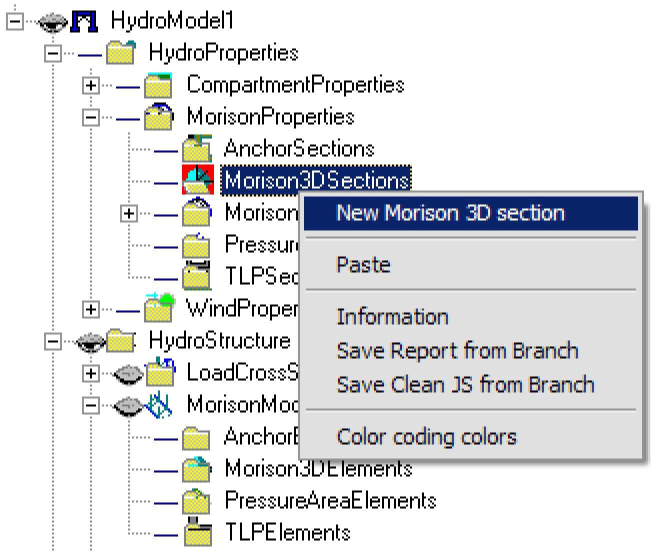
- Length
- Static force (pre-tension)
- Stiffness
- X and Y top offset

5.3.11.7 Morison 3D Elements



Morison 3D elements can be used to produce Morison forces in 3 directions on nodes in the Morison model (Wadam only). A node and a Morison 3D section define a Morison 3D element. In addition a guide node may be specified to change the local coordinate system of the element (see tool tip for explanation).

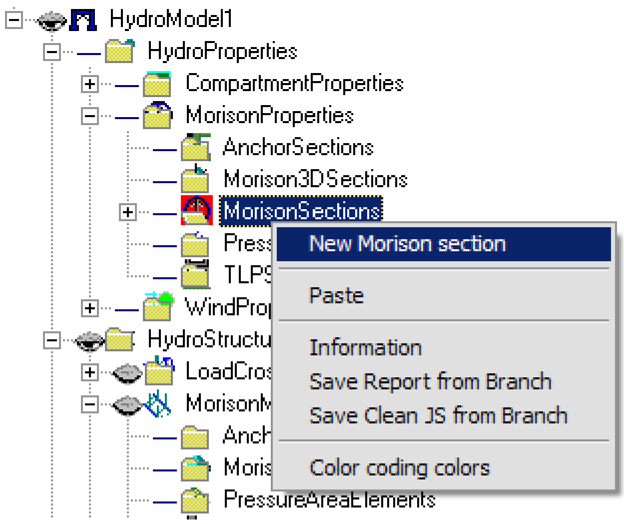
5.3.11.8 Morison 3D Sections

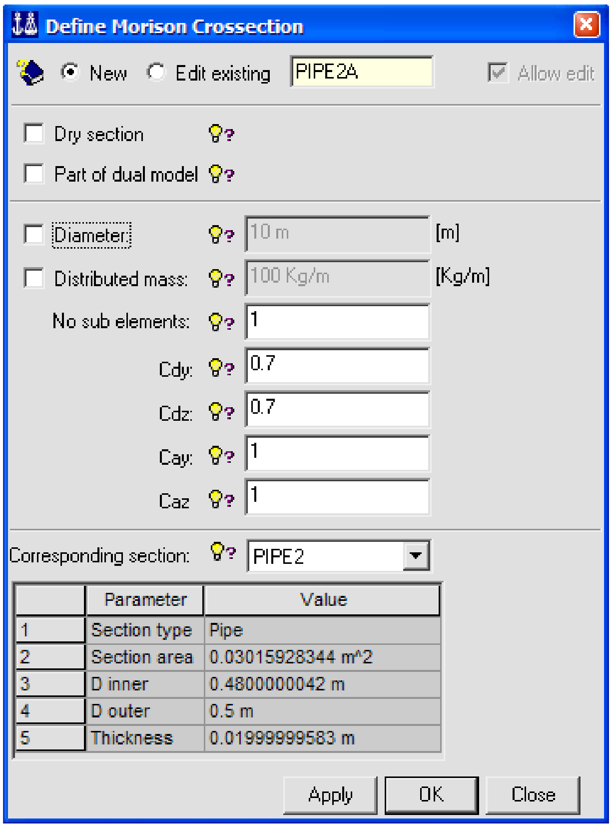


Characteristic common properties of Morison 3D elements including:

- Whether to include buoyancy force on the element
- Whether to include dynamic forces on the element
- Radius
- Mass
- Drag coefficients
- Added mass coefficients

5.3.11.9 Morison (2D) Sections



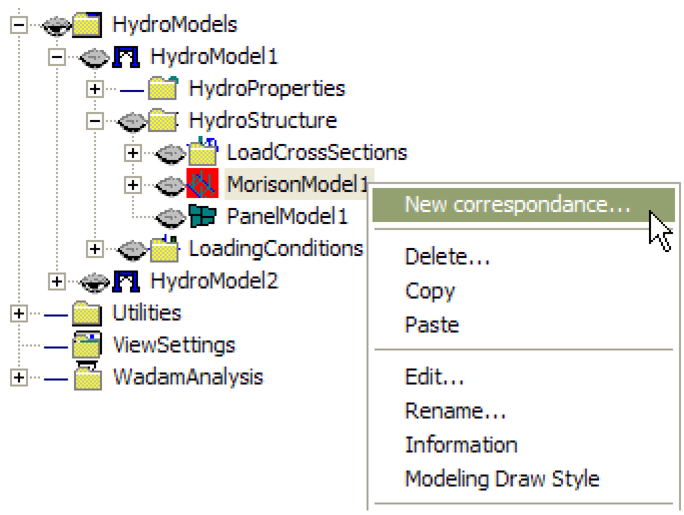


Characteristic common properties of 2D Morison beams including:

- Whether this is a dry Morison section. It is necessary to define dry Morison sections when elements without Morison loading shall receive panel pressures in a dual model.
- Whether elements using this section may be part of a dual representation. This flag is used when an element to panel correspondence is set up. Elements whose section is not part of a dual representation may not be connected to panels.

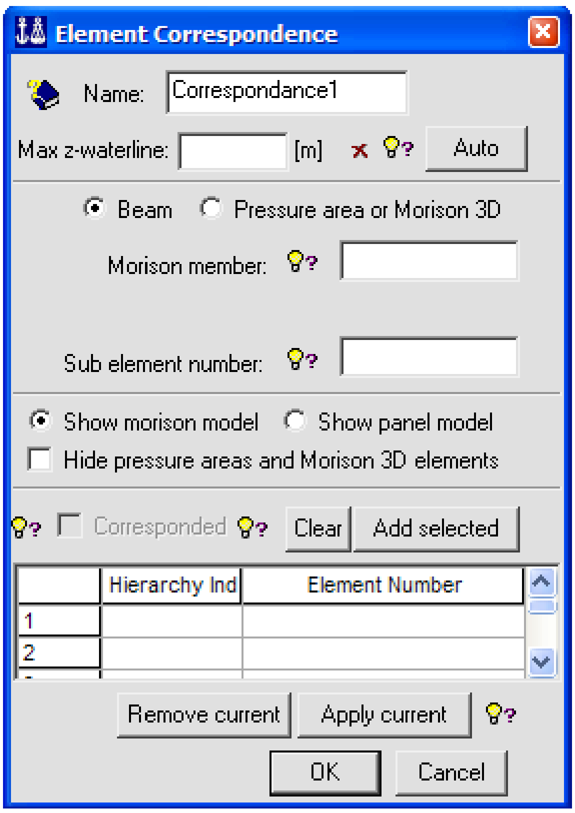
- Hydrodynamic diameter. Should be defined such that the buoyancy area of the cross section is correct. For pipes it is an option to employ the diameter specified on the FEM file.
- Distributed mass. This option is only relevant when the Morison model is used as the mass model and you do not want to use the mass properties defined by the FEM file.
- Number of sub elements. Morison forces are assumed constant over each sub element. In a dual model each sub element should be connected to one or more panels.
- Drag coefficients. Direction of coefficients is explained by tool tip in dialog.
- Added mass coefficients. Direction of coefficients is explained by tool tip in dialog. Added mass force is proportional to $(1+C_M)$.

5.3.11.10 Element Correspondence



All wetted panels in a dual model (Wadam only) must be connected to a Morison sub element, a pressure area element or a Morison 3D element. This serves two purposes:

1. Loads are transferred from the panels and onto the beams/nodes that they are connected to in the correspondence.
2. Hydrostatic and hydrodynamic loads must not be duplicated in the dual parts (i.e. a Morison beam element inside the panel model shall only contribute to the drag part of the dynamic force etc).



Each wetted (meaning that is has a Morison section) beam sub element inside the panel model should be connected to at least one panel.

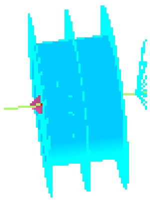
All pressure area elements inside the panel model should also be given correspondence. These may however be given correspondence without connecting them to any panels (so called “empty correspondence”, meaning that the pressure area elements shall not contribute to the dynamic force).

Note that even though a pressure area element shall not contribute to the dynamic force, you still need to define it in a dual model because the static load case in a load transfer run is taken entirely from the Morison model (the panel model has no contribution).

On top of the correspondence dialog there is a button to generate a suggestion for the correspondence table (“Auto”). This suggestion should be visually inspected and verified.

Elements are manually connected to panels by the following steps:

- Position the mouse cursor in the “Morison member” edit control
- Select the relevant beam, pressure area or Morison 3D element in the 3D window (press the left mouse button on the relevant element). Alternatively write the element number/name manually in the control.
- When a beam element has been selected, a sub elements object will be drawn in the 3D window.



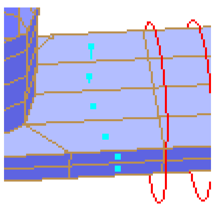
Sub element number is chosen by selecting one of the sub elements.

- Once an element has been chosen the panel model will automatically appear. If the object was selected by manual writing, one has to press the “Show panel model” radio button. The selected sub element is outlined by two circles.
- Select panels graphically in the 3D window (see the “Selection” paragraph).
- Press the “Fill from selected” button to fill the table with the panels that you have selected graphically. Alternatively you may write the panels manually in the table.
- Press “Apply current” to connect the panels in the table to the selected (sub) element.

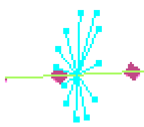
If panels were previously connected to an element when trying to connect them to a new element, the connection to the previous element will be removed.

Use the “Remove current” button to say that a (sub) element shall no longer be connected to any panels.

As the process progresses connected panels will be marked with a point:



In addition lines are drawn from the centre of the relevant sub element and out to the connected panels:



Pressure areas may sometimes make it difficult to see and select beam sub elements. Press the “Hide pressure areas and Morison 3D elements” button to relieve this problem.

It is also worth noting that if one wants to label nodes or beam elements or make other kinds of changes to the modelling draw style of the Morison model (or any other object), one may right click the Morison model in the browser (while the correspondence dialog is up) and choose “Modelling Draw Style”. Here you can choose to visualize the object in the manner that best fits your purpose (see “Draw style Settings” paragraph).

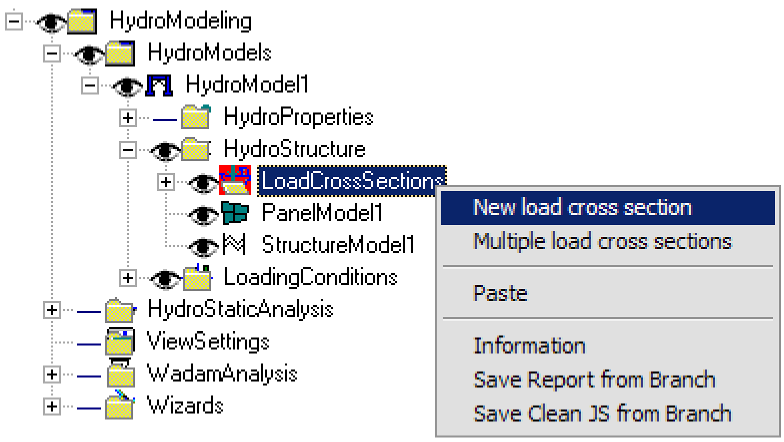
5.3.12 Load Cross Sections

Sectional loads are calculated by integrating load and mass forces up to given cut planes. Load cross sections are defined by a point and a coordinate plane that the cross section is parallel to. Moments are calculated about the given point.

The cross sections may be defined in two ways

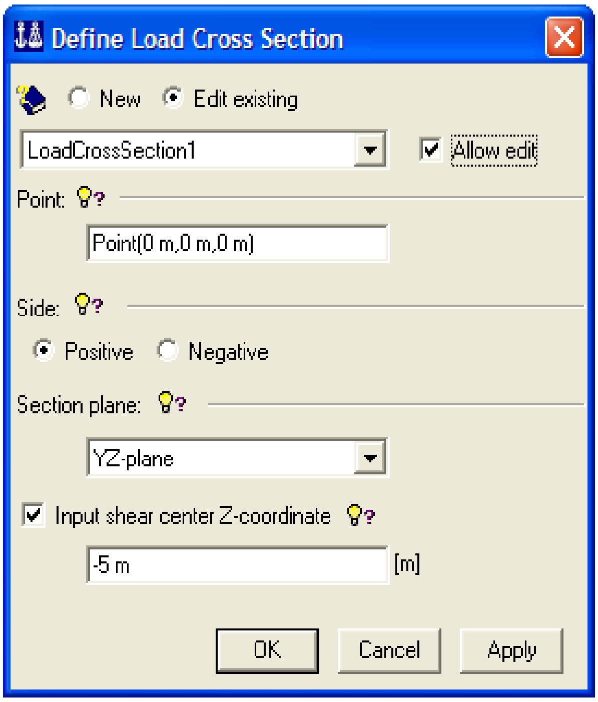
- Single cross sections
- Multiple cross sections

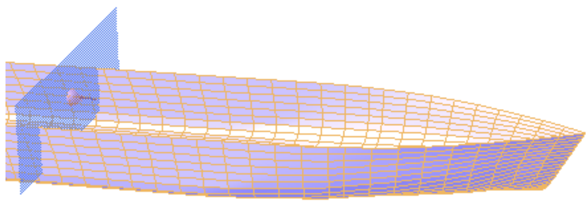
5.3.12.1 Single Cross Sections



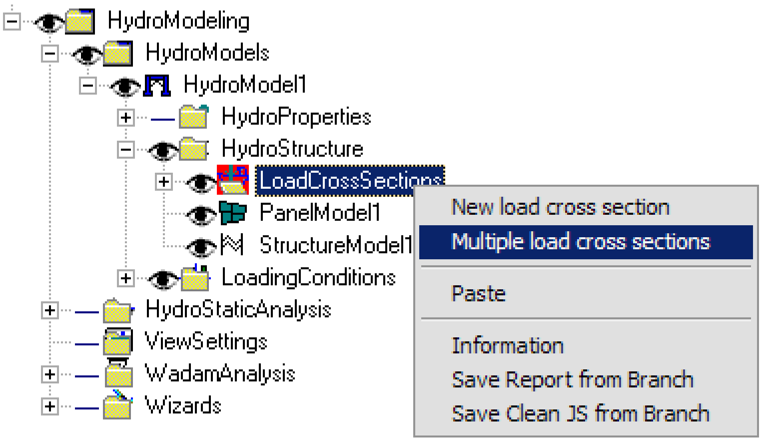
In the following menu, the cross section is defined by giving a point and a plane, in addition to which side of the plane is to be used for the calculation.

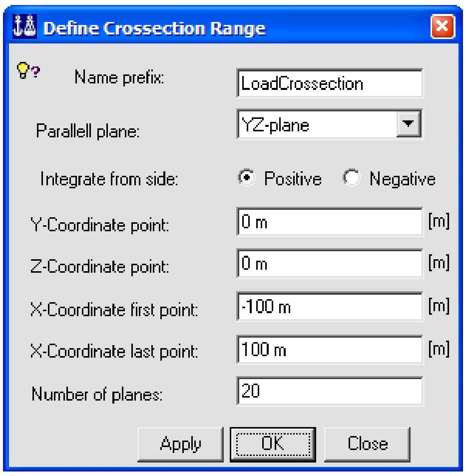
The option for providing a Shear centre coordinate is only relevant for Wasim analysis. If this is given the torsional moment is computed relative to an axis at this z-level, while the bending moments are computed around axes through the specified point.



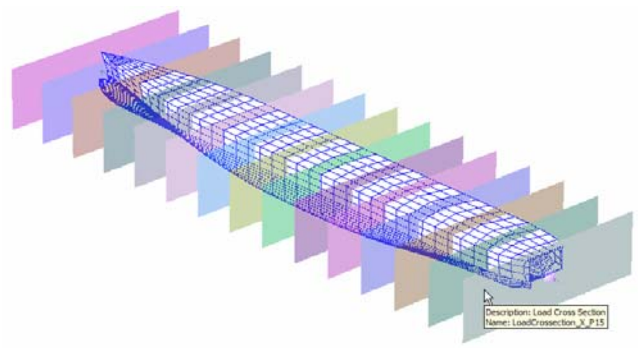


5.3.12.2 Multiple Cross Sections



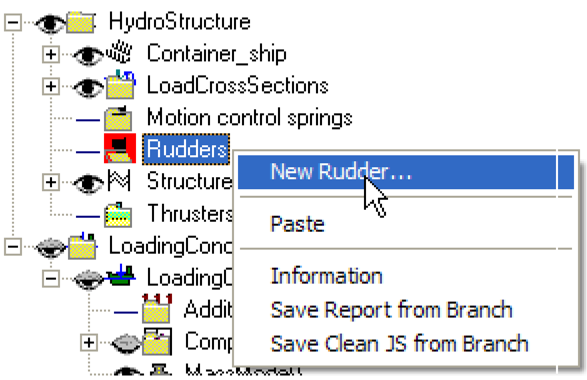


In the following menu, the cross sections are defined by defining a sequence of sections over a length of the structure, in addition to which side of the plane is to be used for the calculation.



Shear centre coordinate can not be given for multiple load cross sections. This parameter must be edited in each section after they have been created.

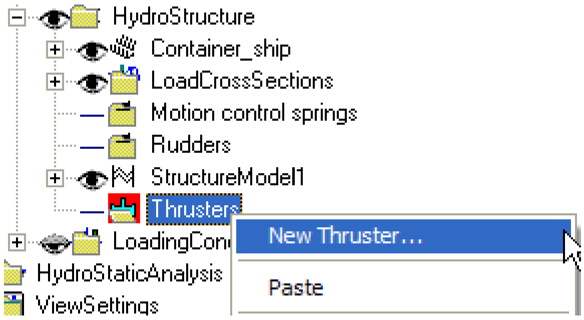
5.3.13 Rudders



Rudders can be used for motion control in Wasim. The input parameters provided do not necessarily have to be physically correct since this should be regarded just as a numerical method for controlling the sway and yaw motion.

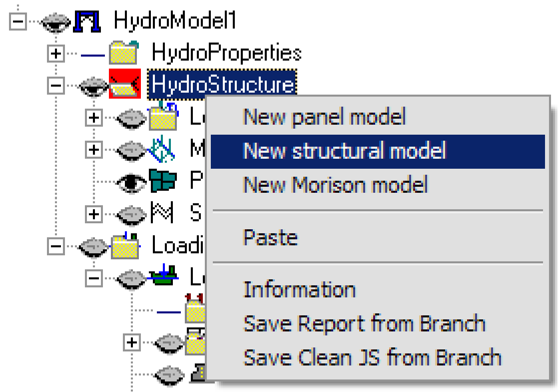
Several rudders can be defined, but only one is used in the analysis. The rudder to be used is selected in the setting for the Wasim analysis.

5.3.14 Thrusters



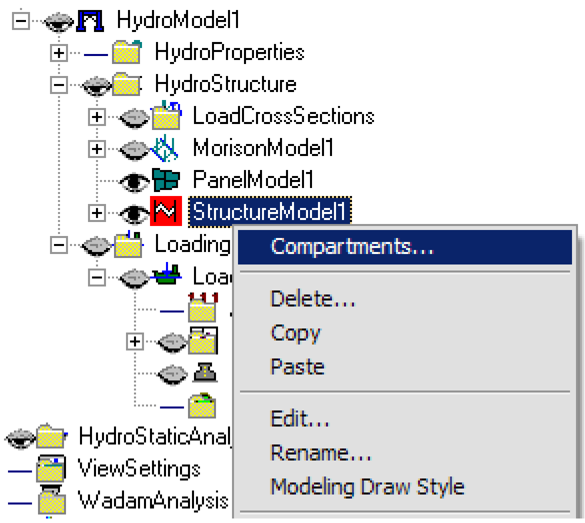
Thrusters are only used in the stability analysis. In this analysis the thrusters force will always give a destabilizing moment.

5.3.15 Structure Model



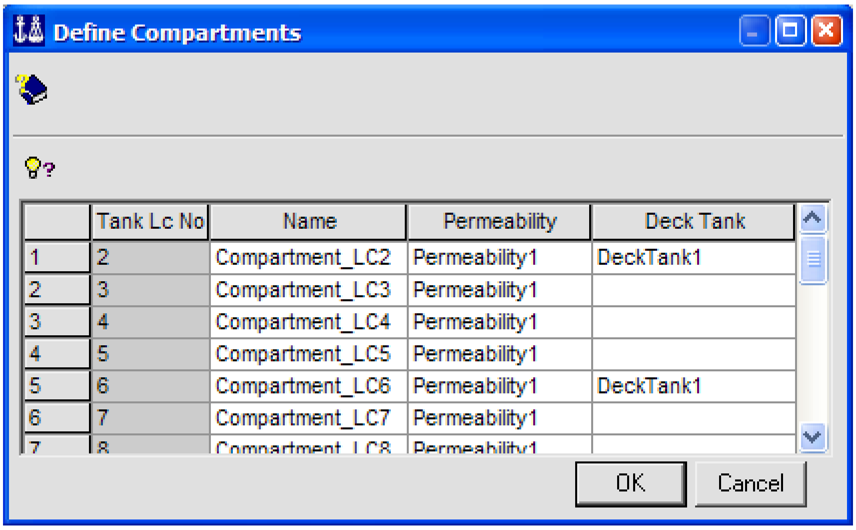
Structure models are defined by a T*.FEM file. (See paragraph about Structure FEM.)

5.3.15.1 Compartments



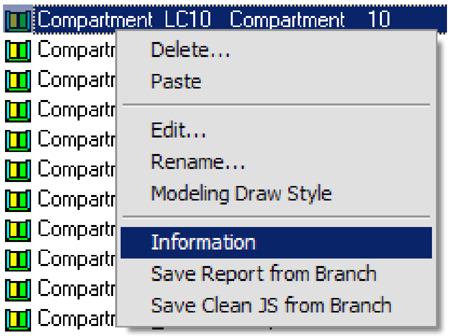
The compartments are defined by definition of wet surfaces/hydro pressure in the structure FEM. All elements inside the compartment must have this definition, starting in load case number 2. (See paragraph about Structure FEM.) In the following menu, each compartment/tank is given appropriate property definitions of permeability and deck tank (independent of the loading condition).

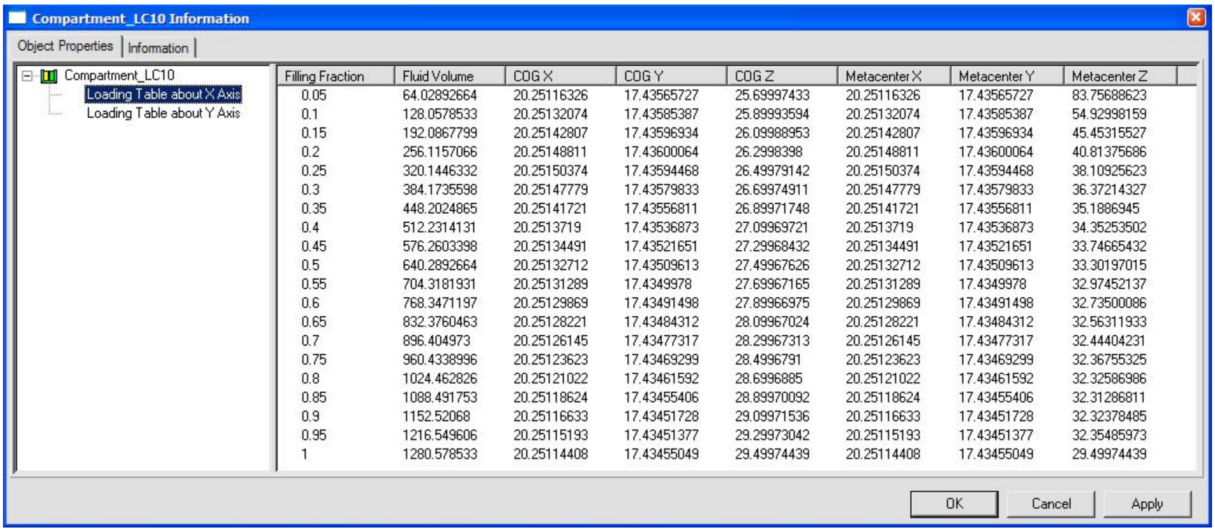
This property may be assigned by using the method "Apply property to selection".



The compartments are also assigned properties in the [Compartment Contents](#) folder under Loading Conditions.

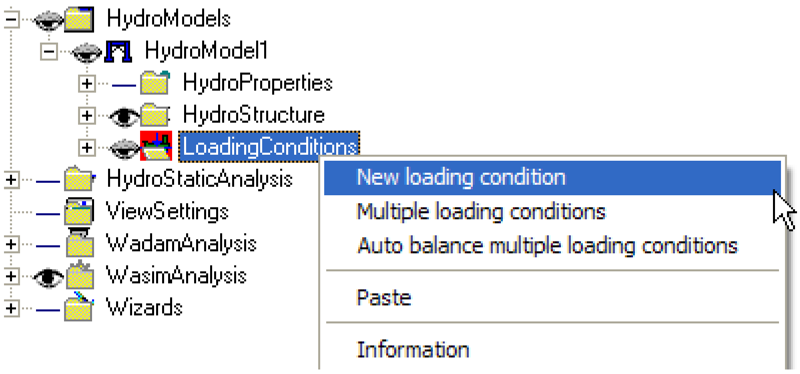
Useful information, such as fluid volume and COG for different filling fractions of the compartments, may also be found by use of the Information option for a specific compartment.





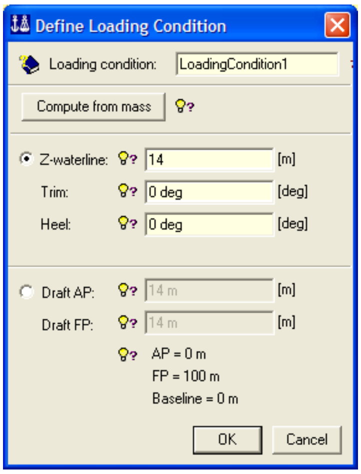
Filling Fraction	Fluid Volume	COG X	COG Y	COG Z	Metacenter X	Metacenter Y	Metacenter Z
0.05	64.02892664	20.25116326	17.43565727	25.69997433	20.25116326	17.43565727	83.75688623
0.1	128.0578533	20.25132074	17.43585387	25.89993594	20.25132074	17.43585387	54.92998159
0.15	192.0867799	20.25142807	17.43596934	26.09988953	20.25142807	17.43596934	45.45315527
0.2	256.1157066	20.25148811	17.43600064	26.2998398	20.25148811	17.43600064	40.81375686
0.25	320.1446332	20.25150374	17.43594468	26.49979142	20.25150374	17.43594468	38.10925623
0.3	384.1735598	20.25147779	17.43579833	26.69974911	20.25147779	17.43579833	36.37214327
0.35	448.2024865	20.25141721	17.43556811	26.89971748	20.25141721	17.43556811	35.1886945
0.4	512.2314131	20.2513719	17.43536873	27.09969721	20.2513719	17.43536873	34.35253502
0.45	576.2603398	20.25134491	17.43521651	27.29968432	20.25134491	17.43521651	33.74665432
0.5	640.2892664	20.25132712	17.43509613	27.49967626	20.25132712	17.43509613	33.30197015
0.55	704.3181931	20.25131289	17.4349978	27.69967165	20.25131289	17.4349978	32.97452137
0.6	768.3471197	20.25129869	17.43491498	27.89966975	20.25129869	17.43491498	32.73500086
0.65	832.3760463	20.25128221	17.43484312	28.09967024	20.25128221	17.43484312	32.56311933
0.7	896.404973	20.25126145	17.43477317	28.29967313	20.25126145	17.43477317	32.44404231
0.75	960.4338996	20.25123623	17.43469299	28.4996791	20.25123623	17.43469299	32.36755325
0.8	1024.462826	20.25121022	17.43461592	28.6996885	20.25121022	17.43461592	32.32586986
0.85	1088.491753	20.25118624	17.43455406	28.89970092	20.25118624	17.43455406	32.31286811
0.9	1152.52068	20.25116633	17.43451728	29.09971536	20.25116633	17.43451728	32.32378485
0.95	1216.549606	20.25115193	17.43451377	29.29973042	20.25115193	17.43451377	32.35485973
1	1280.578533	20.25114408	17.43455049	29.49974439	20.25114408	17.43455049	29.49974439

5.3.16 Single Loading Condition



A loading condition is defined by the z-coordinate of the waterline, trim and heel angles. All data that are typically dependent upon the draught are placed under a loading condition.

The visual representation of a loading condition is the free surface.

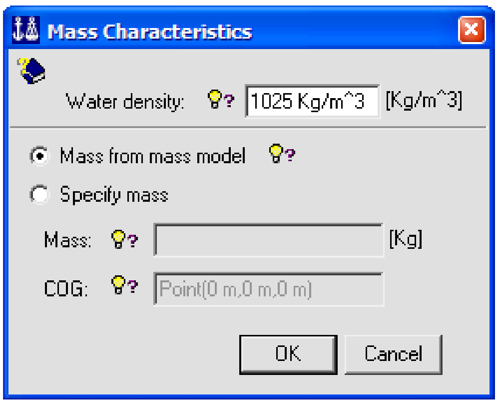


Trim and heel angles are Euler angles performed in the order RX-RY-RZ (i.e. heel before trim) for the input to global transformation. Rotations are done prior to translating the model a magnitude minus "Z-waterline" in the z-direction.

The loading condition can also be defined by draft at AP and FP. If the HydroModel contains a SectionModel, the AP and FP positions are taken from the SectionModel. Otherwise the positions are taken from the HydroModel definition.

The draft at x is defined as the distance from the point (x,0,Baseline) in the input coordinate system to the free surface (measured along a line perpendicular to the free surface).

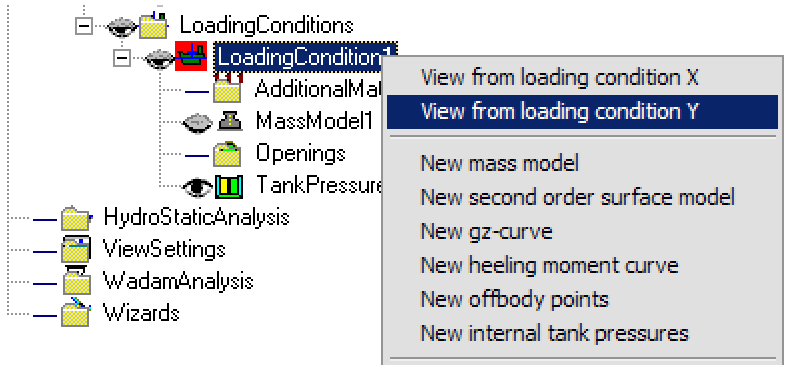
If the equilibrium position is not known, the program may compute it automatically. One must then press the "Compute from mass" button and the following dialog will appear:



HydroD will use the values for "Z-waterline", "Trim" and "Heel" specified in the loading condition dialog as a starting point for an iterative procedure. This means that these values should be specified reasonably close to the equilibrium position prior to pressing the "Compute from mass" button.

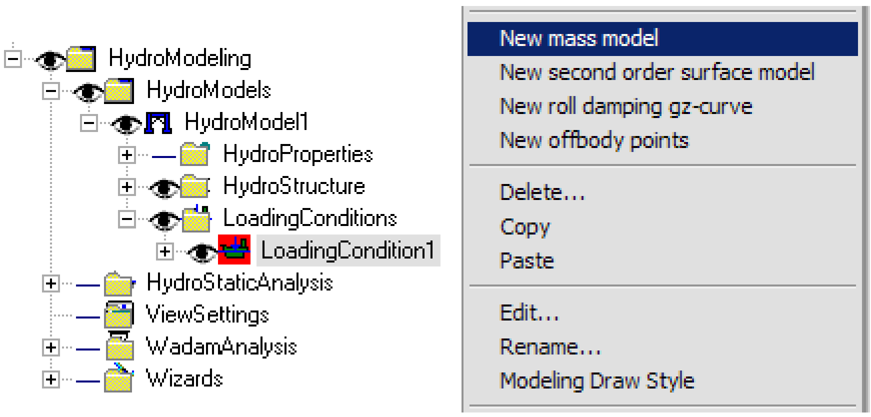
Since the mass model is not specified the first time one defines a loading condition, it is usually necessary to revisit this dialog to compute the equilibrium position after the mass and compartment fillings have been defined.

To ease the 3D window interaction, the view of the model may be rotated automatically according to the resulting trim and heel. The relevant actions ("View from loading condition X/Y") can be accessed by right clicking a loading condition:



To avoid displaying the surface, you have to click on the eye next to the relevant loading condition (in the browser) to switch off the sea surface. Alternatively you can right click the sea surface (in the 3D window), choose “Modelling Draw Style” and click off the “Show” button in the upper left corner.

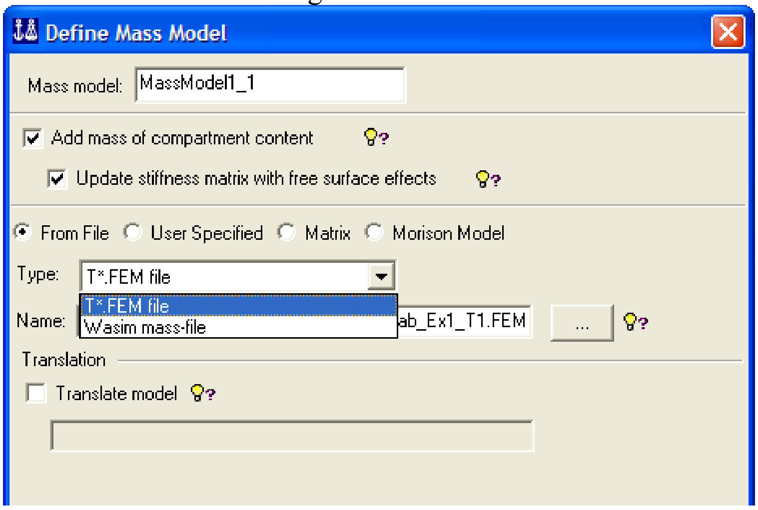
5.3.16.1 Mass Model



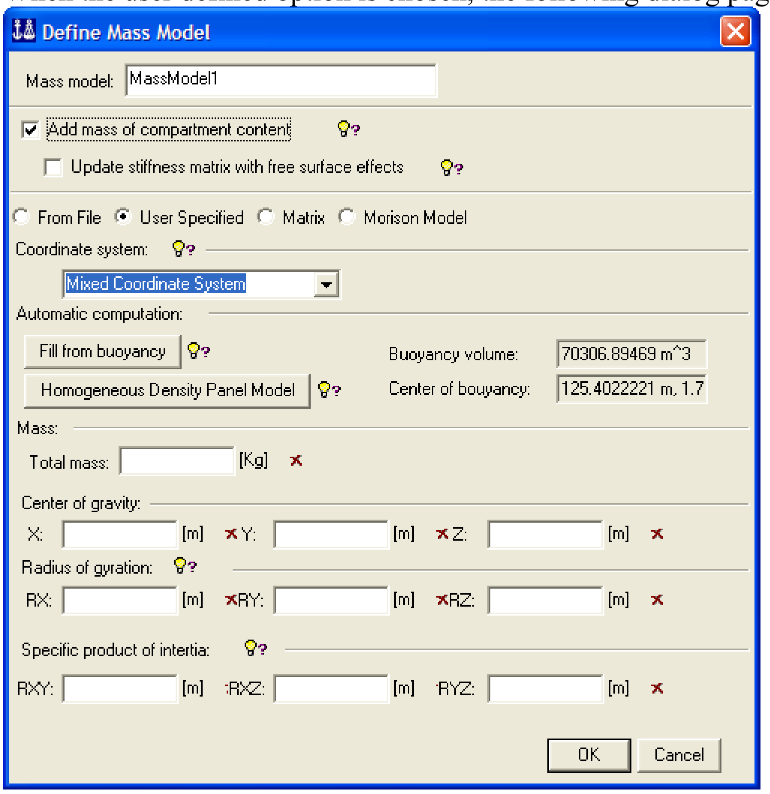
Mass models may be defined in one of four different ways:

1. Mass calculated from a separate mass model file
    a. A mass FEM file (see “Mass FEM” paragraph)
    b. A Wasim mass file, .mas or .mass
2. User defined mass parameters
3. A 6x6 mass matrix
4. Mass given by the Morison model

The mass model file dialogue is as follows:



When the user defined option is chosen, the following dialog page is shown:



By pressing the “Homogeneous Density Panel Model” button, all mass entries will be computed from the panel model. This requires that the panel model is a closed model (i.e. wet surfaces/hydro pressure must be defined also on the deck/super structure). The mass is assumed to be homogeneously distributed within the panel model. It is recommended to press this button before pressing the “Fill from buoyancy” button is pressed.

By pressing the “Fill from buoyancy” button, total mass and centre of gravity is filled in by balancing mass and buoyancy from the panel and/or Morison model.

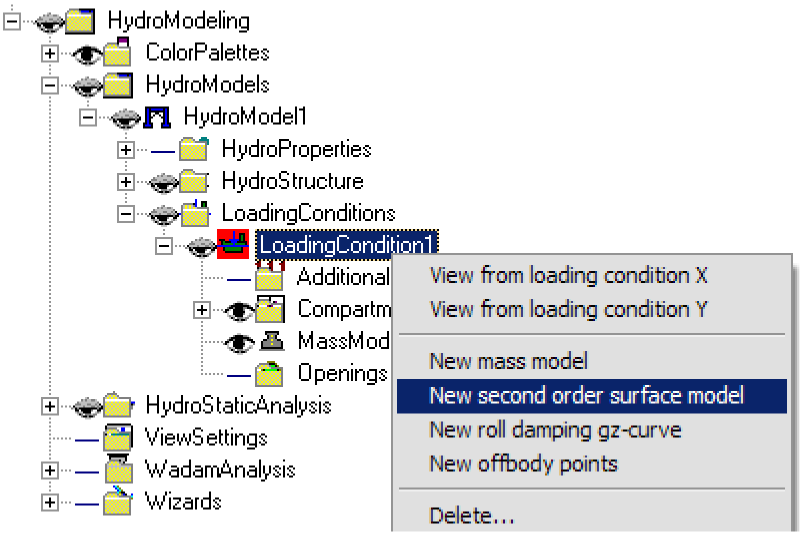
Mass of any defined compartments (in the structural model) may be included. When ticking this box the mass of the internal fluid is added to the mass defined in the Mass model before running the analysis.

The restoring effect due to internal free surfaces can also be transferred to a Wadam or Wasim analysis by ticking this box.

The user may specify the coordinate system in which the user mass is defined. This may be the Input system, the Global system, a “COG centred system” or a mix of both systems. More details are found in the tool tips for this menu. On the listing file written by Wadam the mass data are printed in the mixed system.

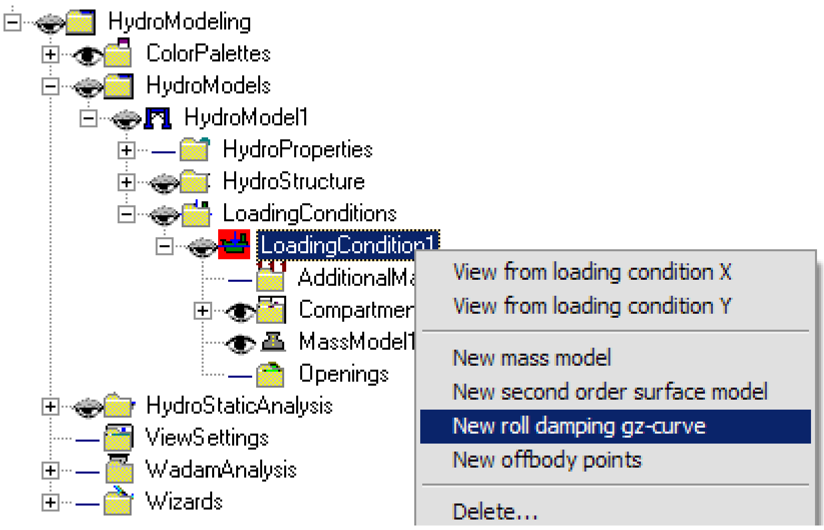
Note that computation of sectional loads in a Wadam or Wasim run requires a distributed mass model, meaning that either the “From File” or “Morison Model” option must be chosen. The “Morison Model” option is only valid for Wadam.

5.3.16.2 Second Order Surface Model



If second order results are to be calculated, a second order free surface model is required. This is defined by a second order free surface FEM (see “Second Order Free Surface FEM” paragraph).

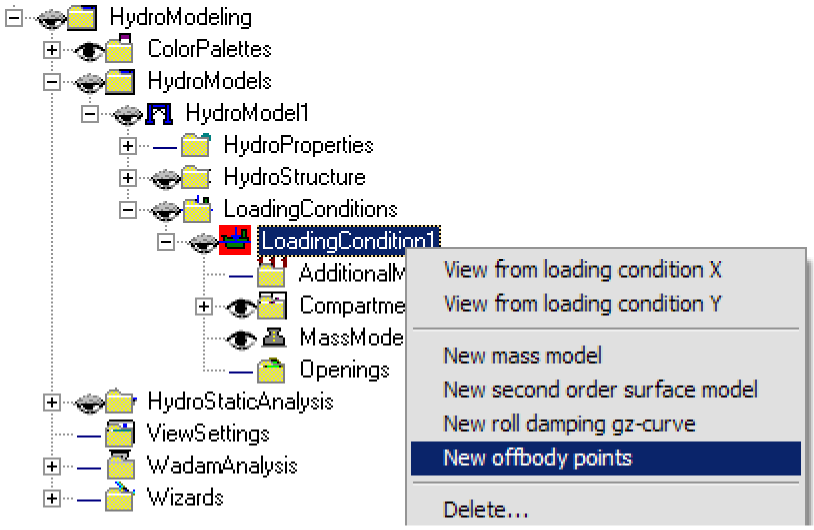
5.3.16.3 GZ-Curve



HydroD can compute GZ-curves. However, when GZ-curves shall be used in viscous roll damping calculations, they must be given as manual table input on the loading conditions. If a hydrostatics and stability analysis has been run, the relevant data can be copied from the stability results and pasted into

the definition dialog of the GZ-curve (save a report in XML-format and copy the data from Excel and into the table). See Wadam user manual for details on how the GZ-curves are employed.

5.3.16.4 Offbody Points



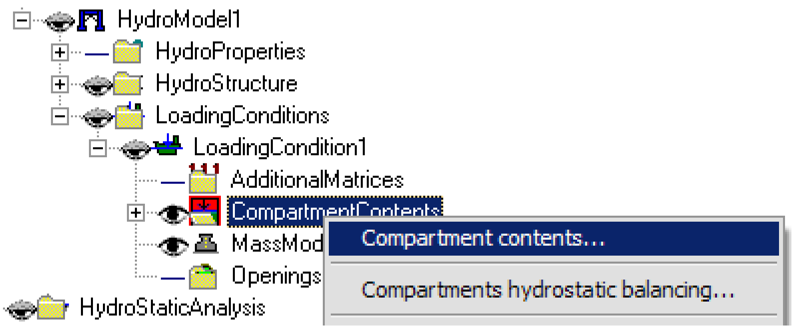
Offbody points are typically used for gap calculations or free surface wave elevation animations. They may be defined in one of three different ways:

1. An offbody points FEM (see Offbody Points FEM)
2. An automatically generated offbody points FEM. User only specifies a min and max point in the XY-plane and the number of elements in each direction.
3. Individual points (X, Y and Z- coordinate of each point), for later computation of air gap etc.

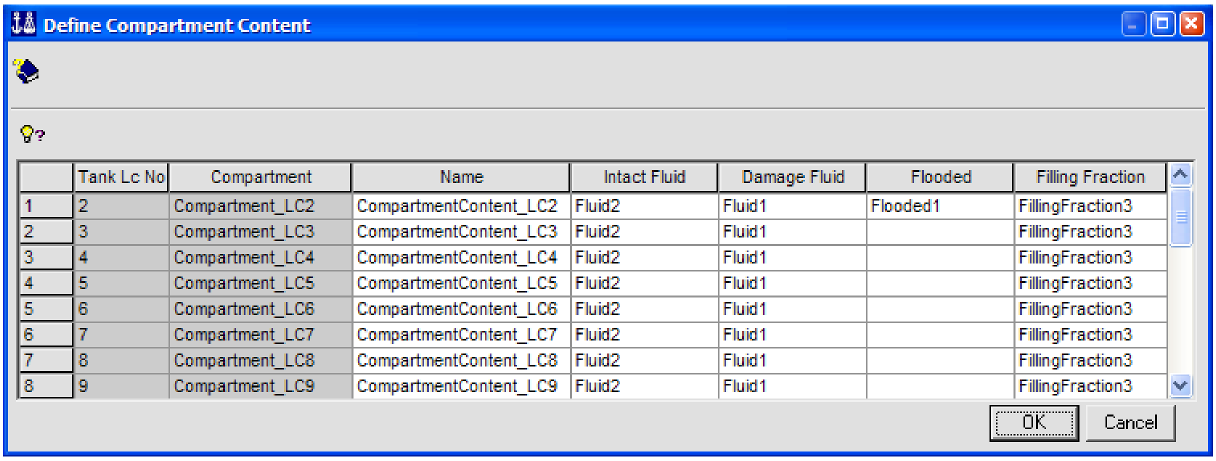
For Wasim also the wave elevations will be output for the offbody points with z-value equal to zero.

5.3.16.5 Compartment Contents

Compartments are defined in the structure model. Some properties are assigned to the compartments in the Structure Model folder (independent of the loading condition) and other properties are assigned to the compartment contents.



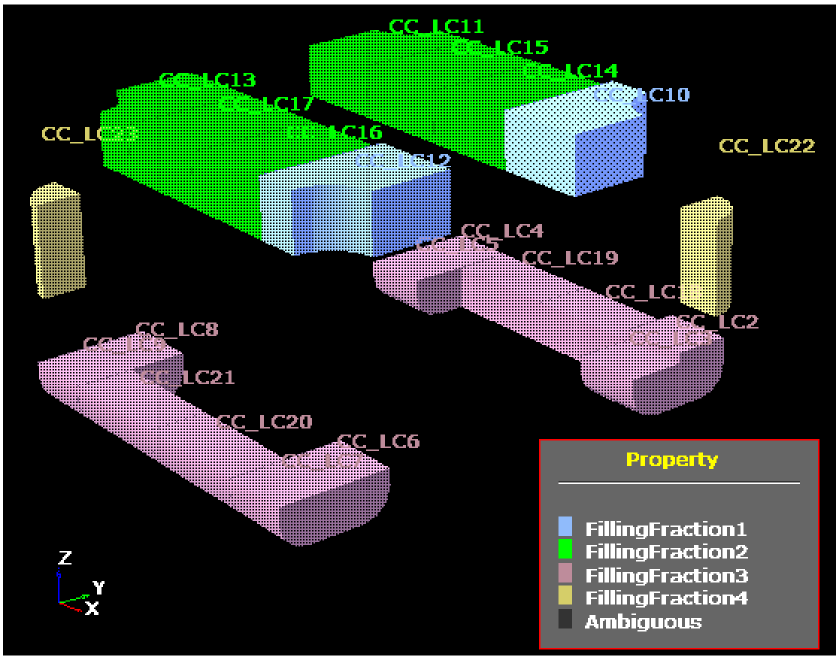
A filling fraction property is assigned to each compartment. A compartment may also be defined as flooded, to simulate a damage condition. A flooded compartment filling is always aligned with the sea surface level. (See also the paragraphs about Compartments and Structure FEM.)



The fluid density is assigned as a fluid property compartment by compartment, including possible definitions of different fluid densities as intact or damaged,.

Compartments are visualized in the 3D window with one colour for each compartment (click the eye). To see the internal of the compartments it may be useful to employ clipping planes (see “Graphical Interaction” section).

The compartments may also be displayed as colour coded according to e.g. the filling fraction.

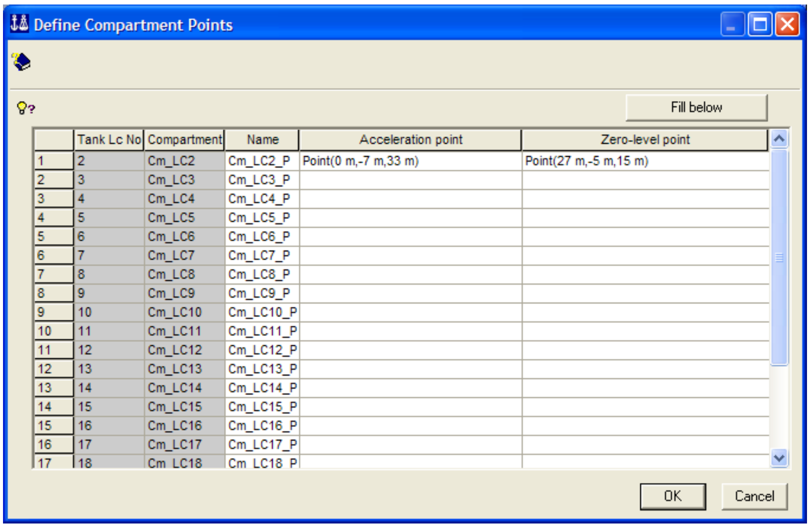


5.3.16.6 Compartment points

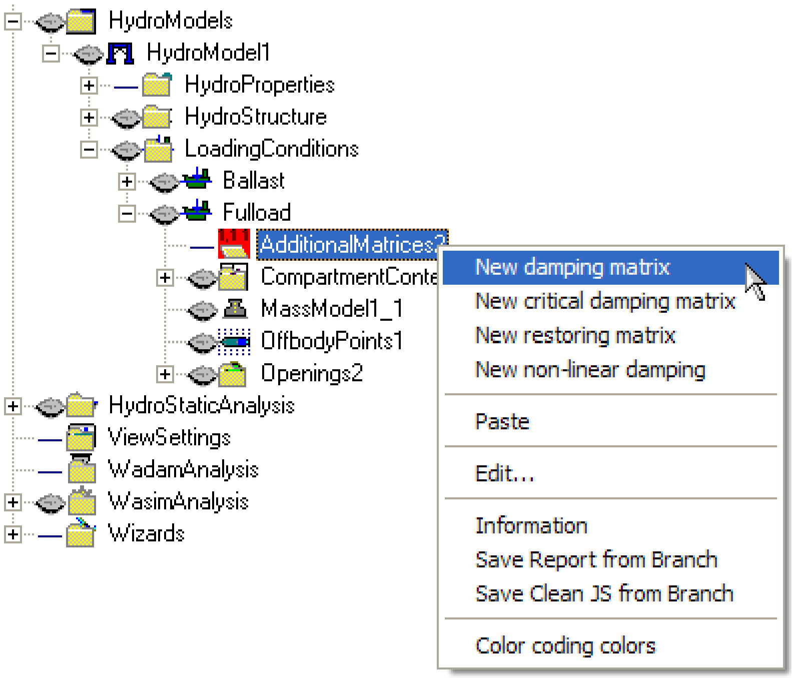
The compartment acceleration and zero level points are defined in “Compartment points” dialog in the loading conditions.

Both acceleration and zero level points are defined in the input coordinate system. The point coordinates should be defined using syntax like “Point(4m, -2.5m, 6m)” or “4 -2.5 6” (with spaces as separators).

The compartment points are currently not visualized in the 3D window.



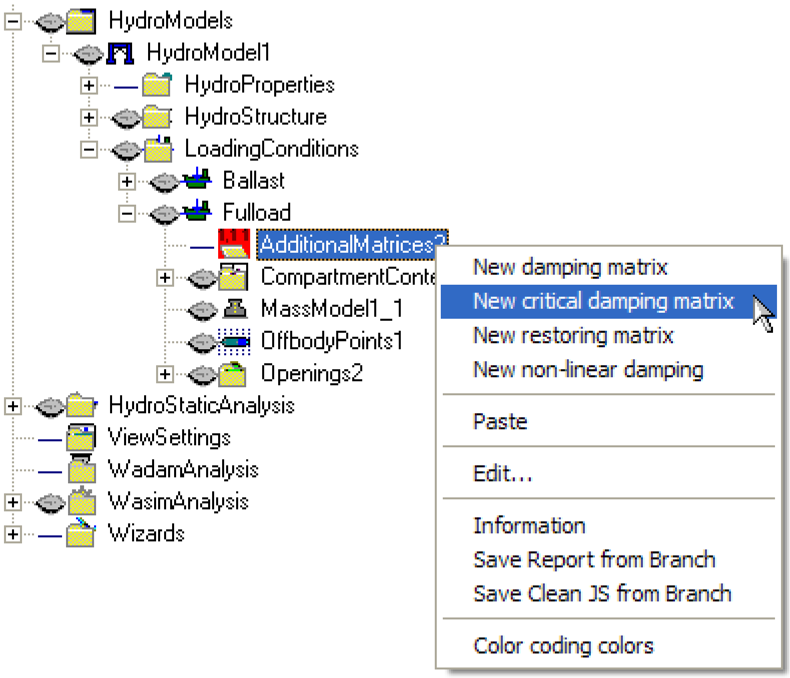
5.3.16.7 Damping Matrix



Additional damping may be defined in the form of a 6x6 matrix. The user may choose whether damping and critical damping matrices shall be added to or replace the damping matrices calculated by Wadam. One damping matrix may be employed for all frequencies, or it may be attached to one period/frequency/wave length. If damping matrices are attached to period/frequency/wave lengths, the damping matrix closest to the relevant period/frequency/wave length in the analysis will be employed.

Note that additional damping matrices may create erroneous sectional loads and imbalance in load transfer.

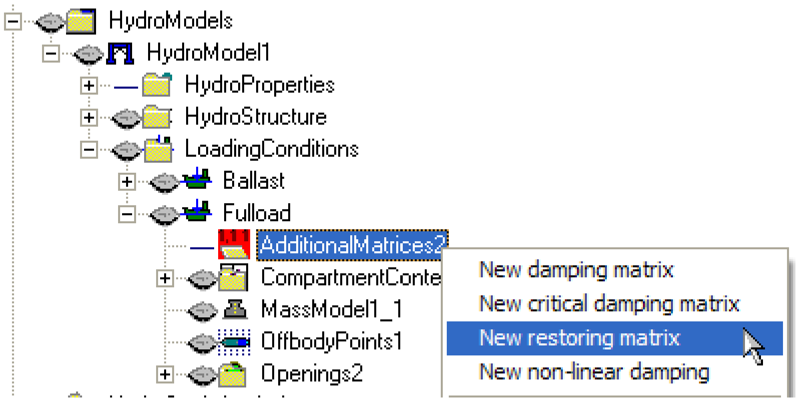
5.3.16.8 Critical Damping Matrix



Additional damping may be defined in the form of fractions of critical damping. Only the diagonal elements of the matrix can be given as input. The user may choose whether damping and critical damping matrices shall be added to or replace the damping matrices calculated by Wadam.

Note that additional damping matrices may create erroneous sectional loads and imbalance in load transfer.

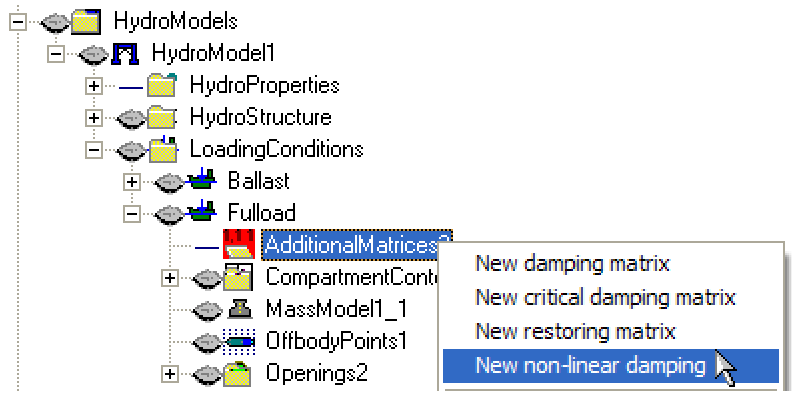
5.3.16.9 Restoring Matrix



An additional 6x6 restoring matrix may be defined. This will be added to the restoring matrix calculated by Wadam. The restoring matrix should be defined in the input coordinate system.

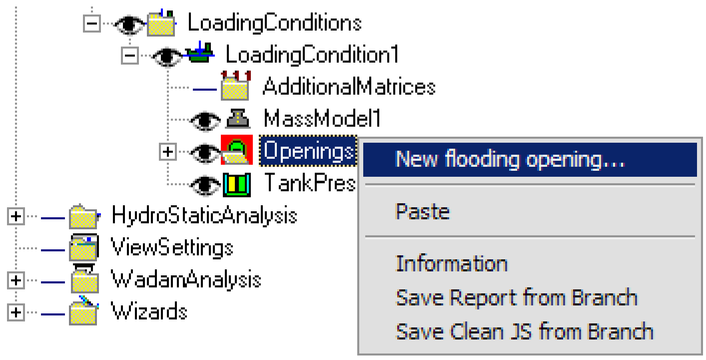
Note that restoring matrices may create erroneous sectional loads and imbalance in load transfer.

5.3.16.10 Non-linear damping



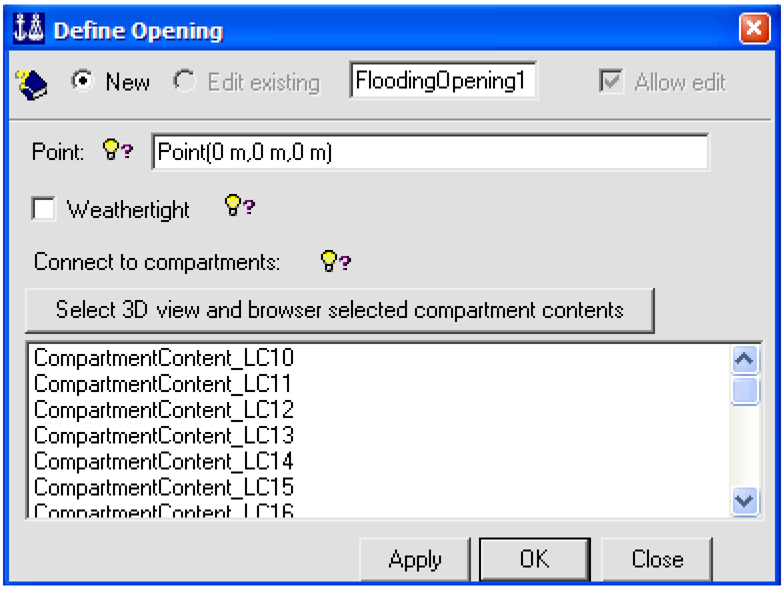
For Wasim a non-linear damping coefficient can be defined for roll and pitch. The non-linear roll damping coefficient is also used for Wadam.

5.3.16.11 Flooding Openings



One or more flooding openings may be defined. These openings will be used in the stability calculation. The distance between the waterline and the opening will be computed and displayed for different heel angles.

The opening is defined by giving a coordinate or by clicking on the wanted point of the model.

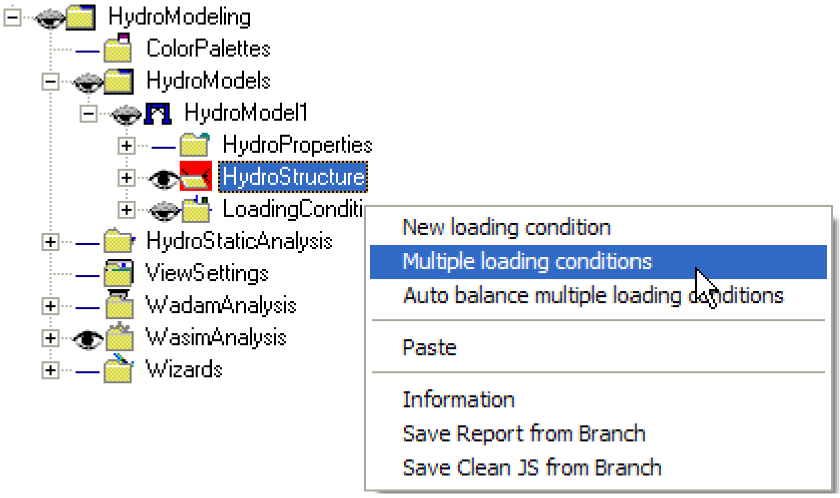


An opening may be defined as Weather tight only, for use in damage stability rule checks.

An opening may be connected to a compartment, for use in damage stability rule checks.

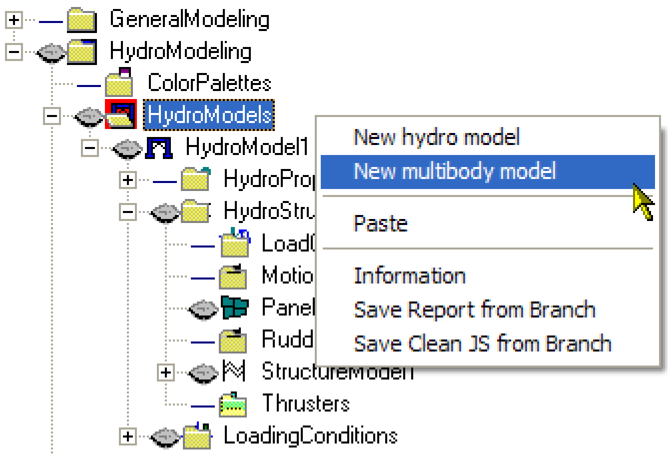
A connected compartment will become flooded when the opening is submerged. This will lead to steps in the computed GZ curve.

5.3.17 Multiple Loading Conditions

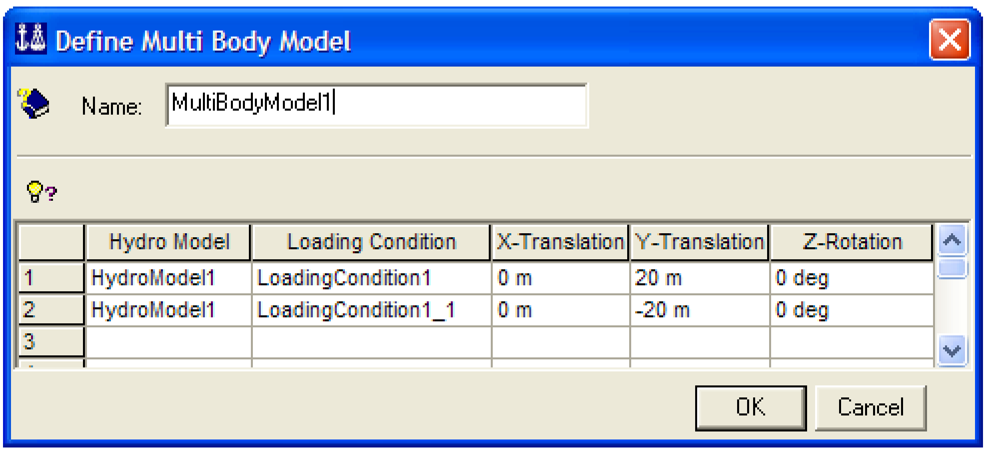


It is also possible to define a sequence of loading conditions in one single operation. The conditions are created based on a template condition, which has to be established before creating multiple conditions. A limitation is that the template condition (and the new conditions) must have a User defined mass model.

5.3.18 Multi-body model



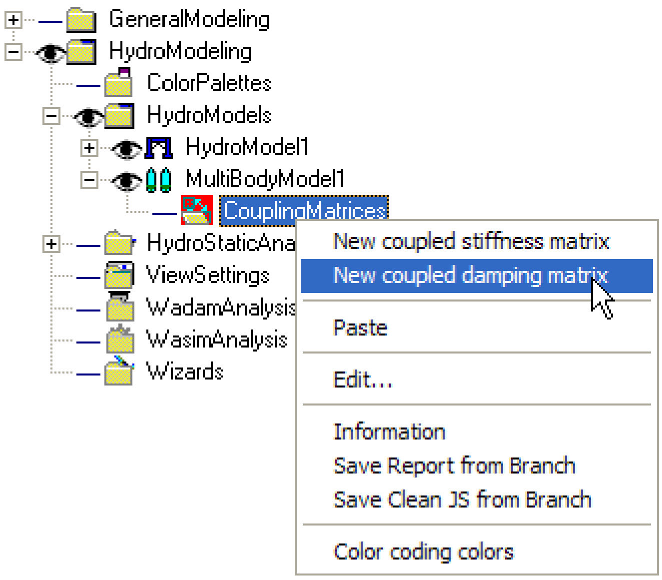
A multi-body analysis in Wadam is based on existing Hydro (panel) models.



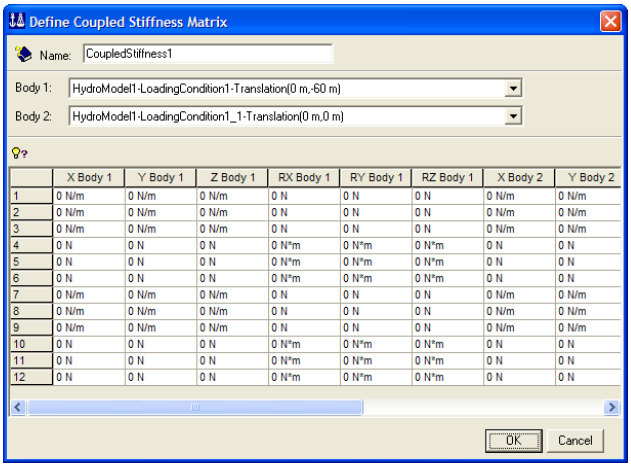
One or more Hydro models may be used, together with one or more loading conditions. Horizontal translations and a rotation may be defined to position the models correctly.

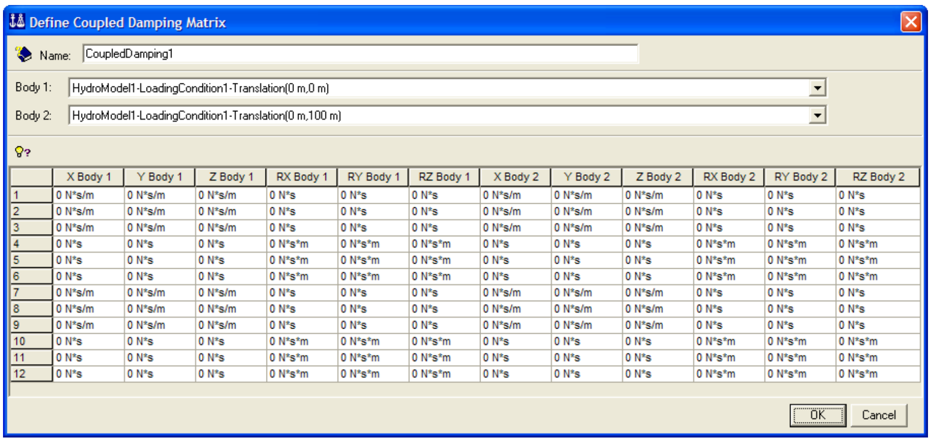
The description of a multi-body analysis in the user manual for Wadam describes three coordinate systems, the input system, the body system and the global system. When running from HydroD, the body system is positioned in the water line at the horizontal position of the model. The results from the multi-body models are reported separately, in the body system for each model.

5.3.18.1 Multi-body stiffness matrix



Coupled damping or stiffness matrices may be defined for the interaction of two bodies.



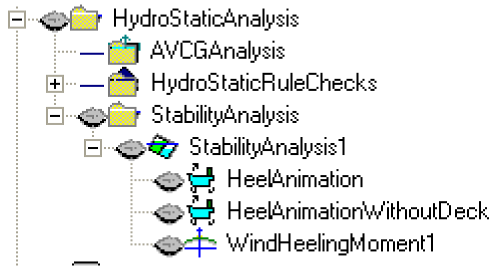


5.4 Analysis

There are three types of analyses (each of which there may be many instances):

- Hydrostatic analysis, run within HydroD
- Hydrodynamic analysis, run by Wadam
- Hydrodynamic analysis, run by Wasim

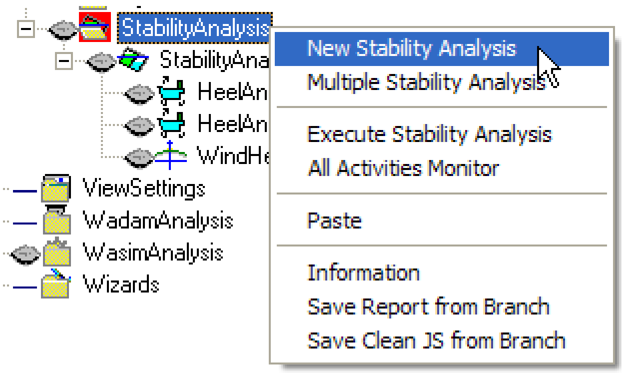
5.4.1 Hydrostatic Analysis

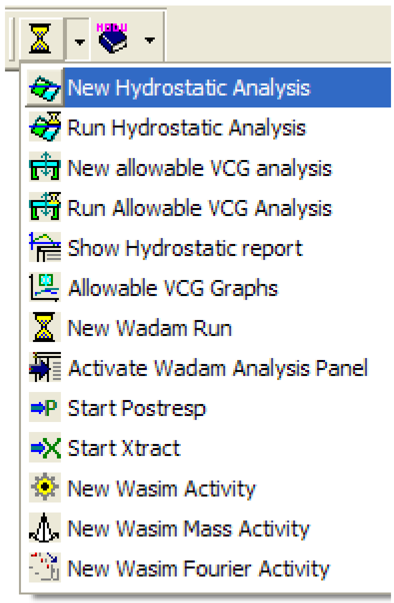


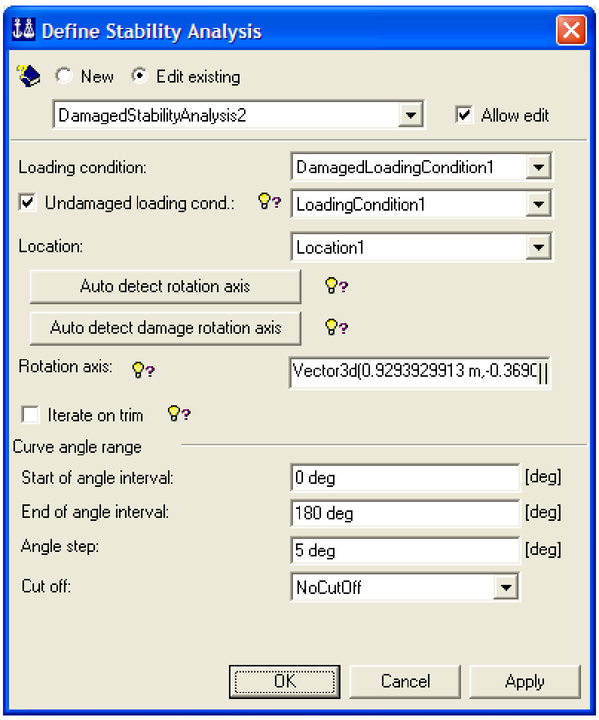
The Hydrostatic Analysis folder contains all the Hydrostatic runs.

5.4.1.1 Single Stability Analysis

A new analysis may be defined by right clicking the browser, or by use of the toolbar.







A Hydrostatic Analysis contains all the information required to execute the analysis. It holds a location, a loading condition and a rotation axis. The hydro model is defined from the loading condition.

For damaged loading conditions one should also specify the corresponding undamaged loading condition. This will be used in rule checks and for the auto detection of the rotation axis.

The "Auto detect rotation axis" button can be used to detect the axis of rotation with the lowest initial

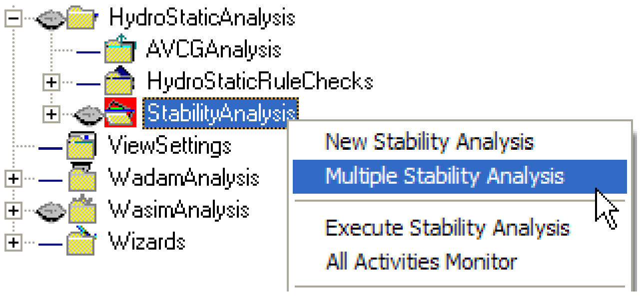
metacentric height (including free surface effects). For damaged loading conditions the “Auto detect damage rotation axis” button returns the axis of rotation from intact to damaged equilibrium position.

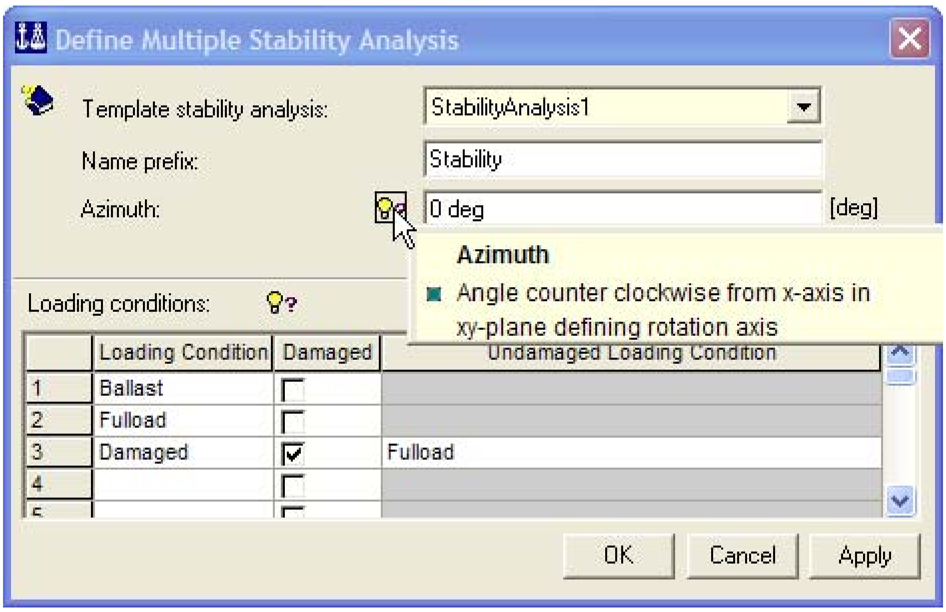
The analysis is performed for a given angle interval and a given angle step.

5.4.1.2 Multiple stability analysis

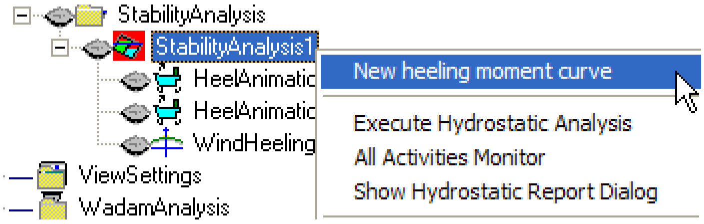
It is possible to define many similar stability analyses in the same command by using the Multiple stability analysis option. Before using this a template stability analysis must be created.

For the damaged conditions the corresponding intact conditions must be specified.

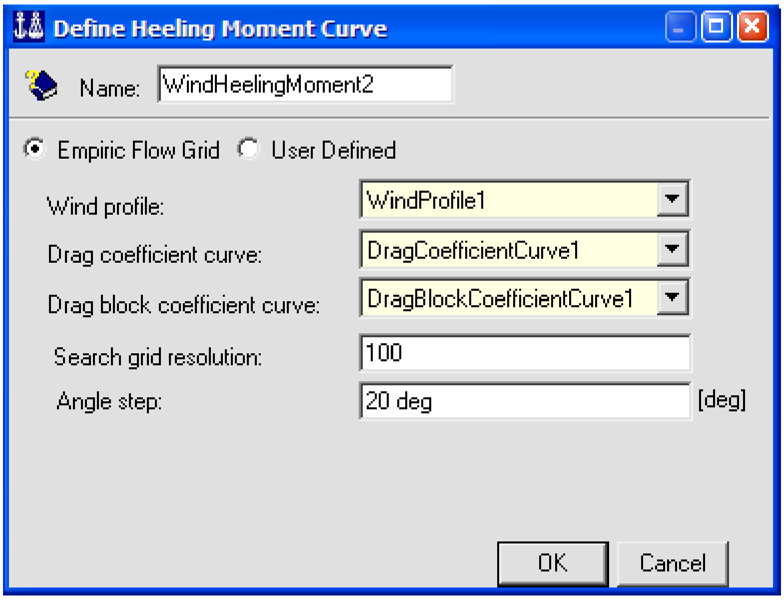




5.4.1.3 Wind Heeling Moment

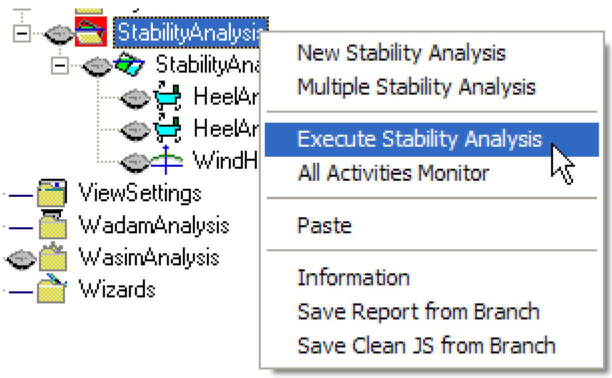


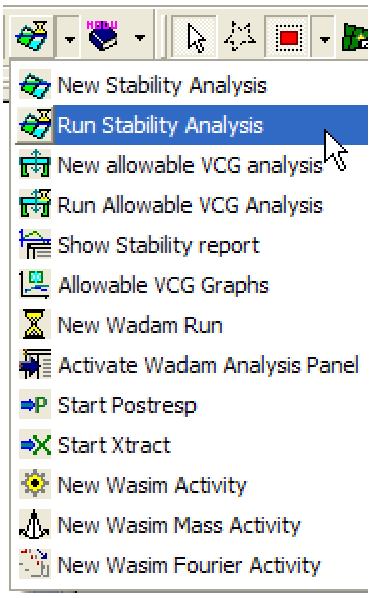
A wind heeling moment curve may be computed, or optionally given directly. The computation is based on the wind profiles and wind properties (drag coefficients) defined in the database.



5.4.1.4 Execute Stability Analysis

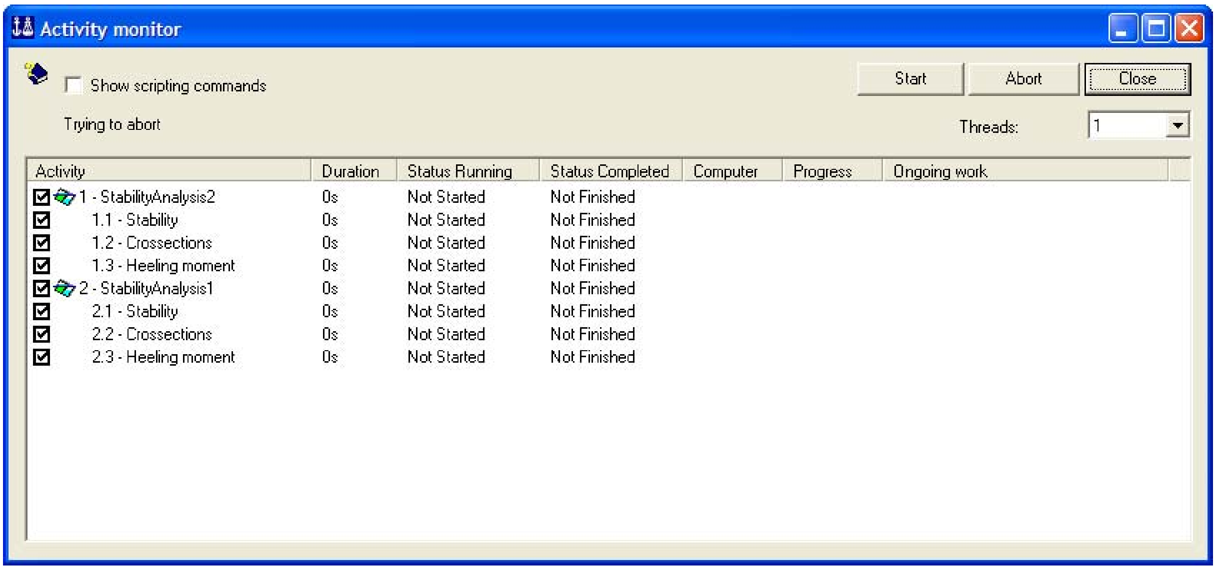
Once the run objects have been defined, you may go into the analysis panel. This can be accessed by right clicking the HydroStaticAnalysis folder. Select either Execute Hydrostatic Analysis or All Activities Monitor:





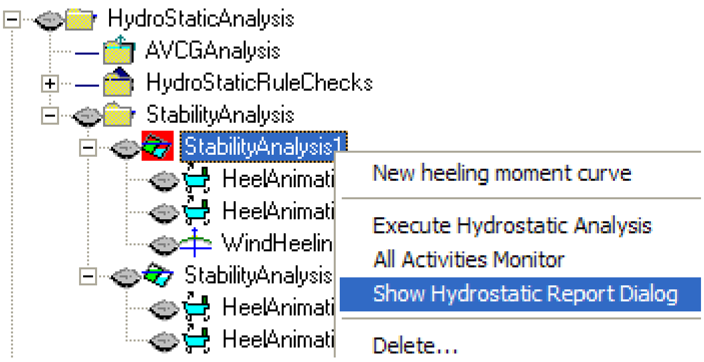
Alternatively use the toolbar button to activate the Analysis Panel.

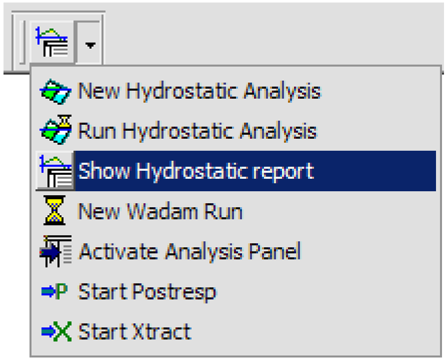
The Activity Monitor includes any defined hydrostatic analyses, and may look as follows:



5.4.1.5 Hydrostatic Results Report

Once a run is completed, you can start looking at the computed results from the browser or from the toolbar button:

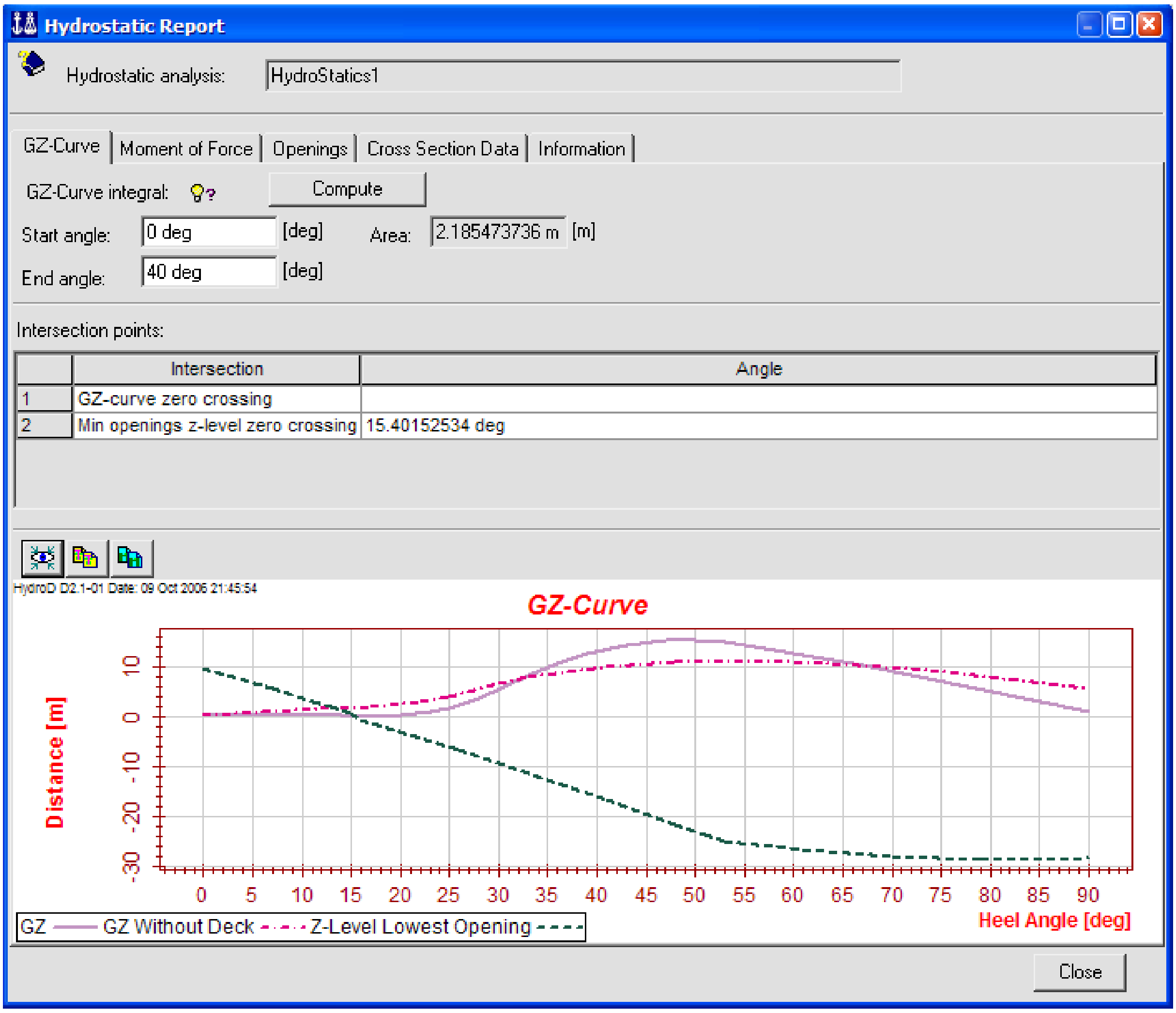




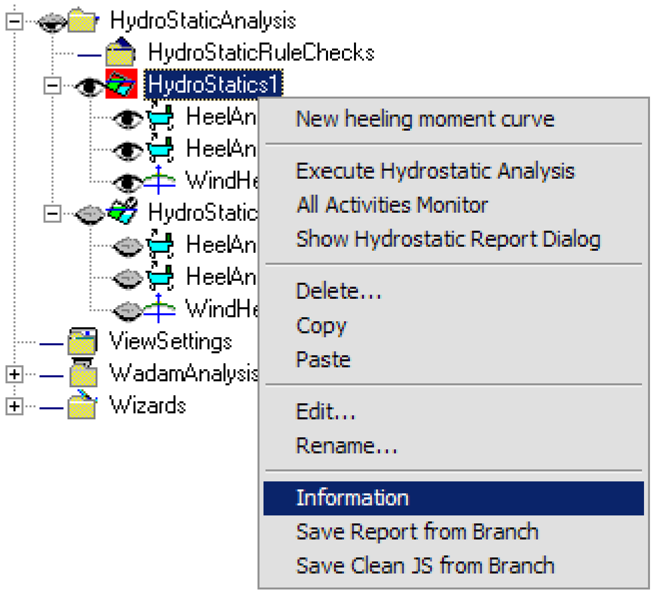
The Hydrostatic Report presents all the calculated results in different tabs of a separate window.

- GZ-curve – displays and prints the computed GZ curve, with and without contribution from compartments in the deck. The vertical level of the lowest opening is also displayed.
- Moment of Force – displays and prints the righting moment curve and heeling moment curve (if defined), including computation of the integrals of the curves for different heel angles
- Openings – displays and prints zero crossing points and the computed distance to the waterline for different heel angles
- Cross Section Data – displays and prints cross sectional loads, forces and moments, for the still water condition, like the longitudinal bending moment. These forces are given in the global coordinate system, i.e. there may be rotation (heel/trim) of the input system.
- Information – prints different results regarding mass and buoyancy calculations, including the metacentric height, and detailed information for defined compartments. This information is given with respect to the input coordinate system.

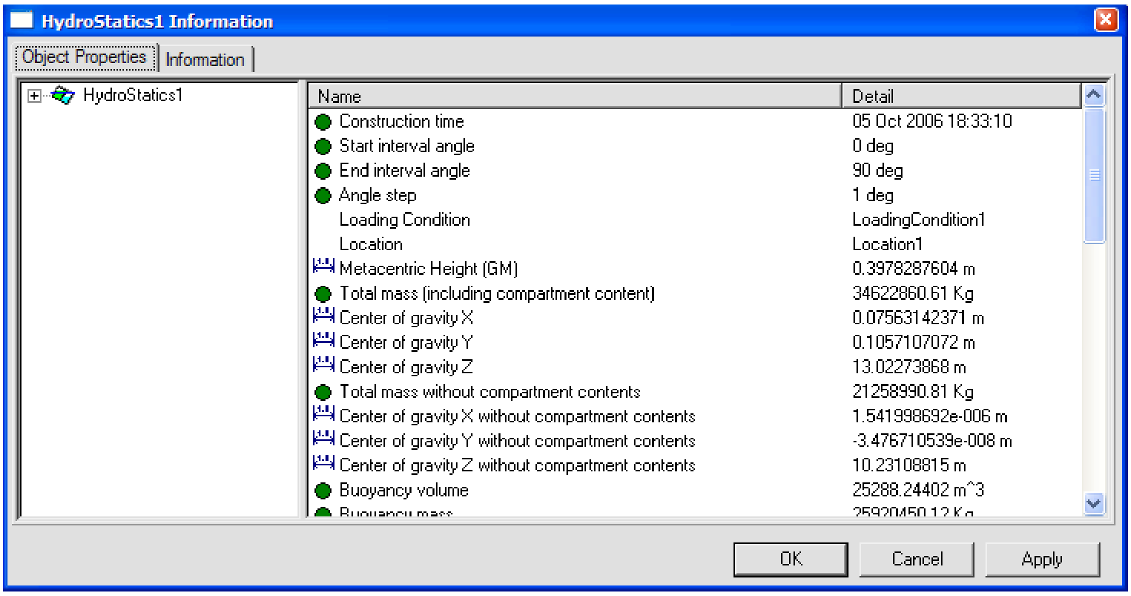
The curves may be copied to plot files. Special settings may be given by right-clicking in the curve window. The first page, for GZ curve results, is shown here:



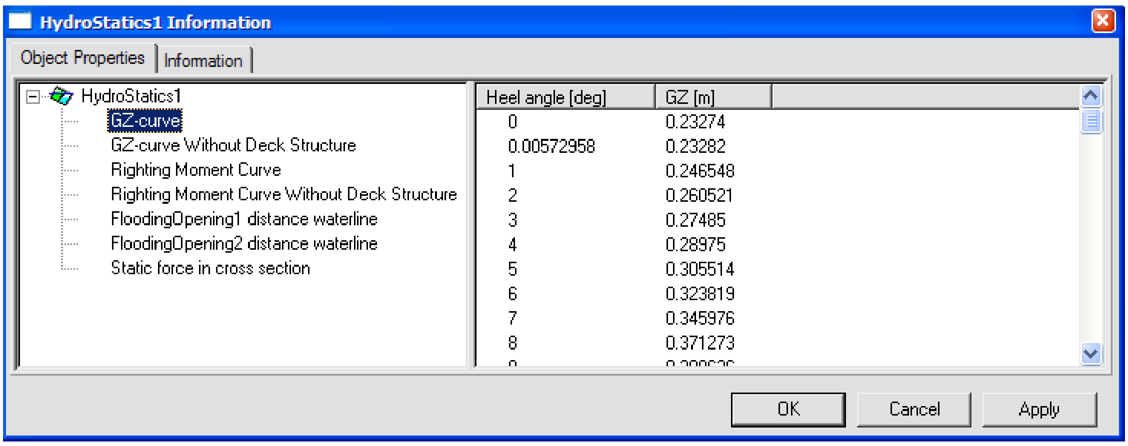
5.4.1.6 Hydrostatic Results Information



The results are also available in another format by selecting Information:



By expanding the browser in the Information window, detailed information on GZ curves etc. is given:

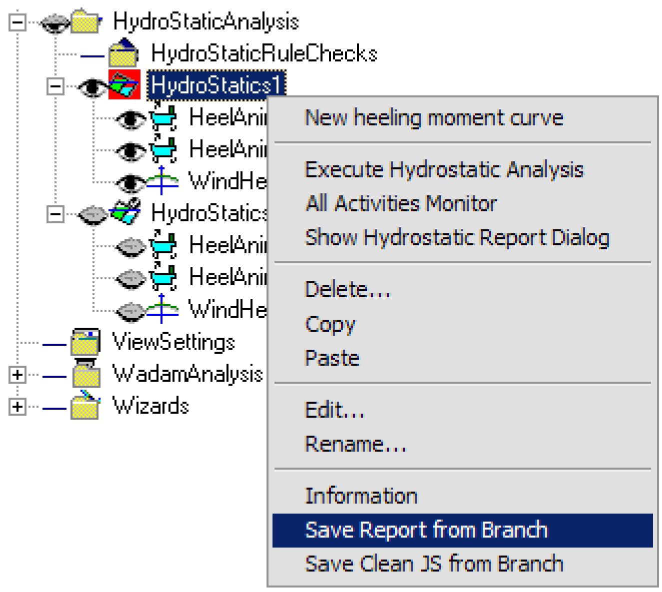


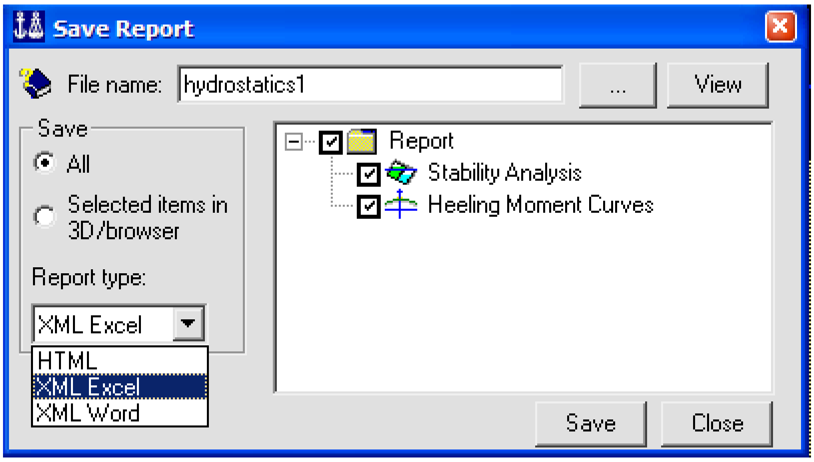
5.4.1.7 Results Report on File

The results may also be printed to a file; the options are HTML format or XML format for either Word or Excel.

After having saved the report, it may be opened by clicking the 'View' button. This option is only available with the HTML format.

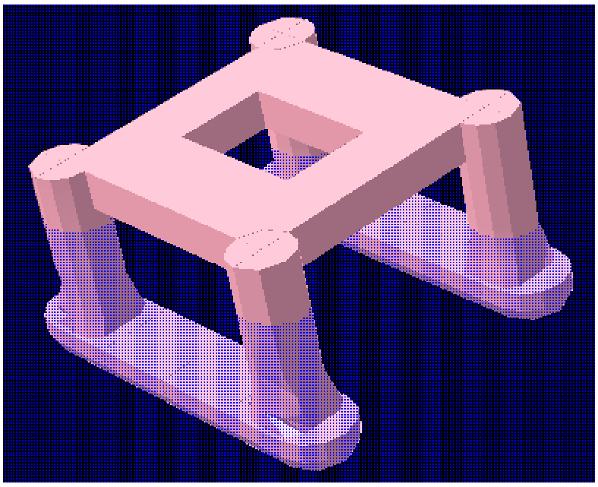
The report file contains all the details of the report, except the figures.

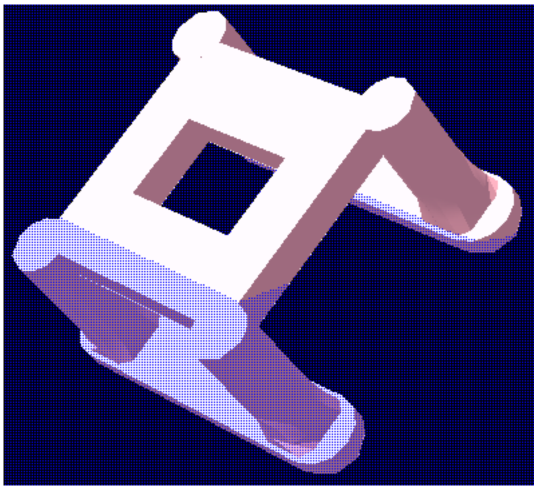




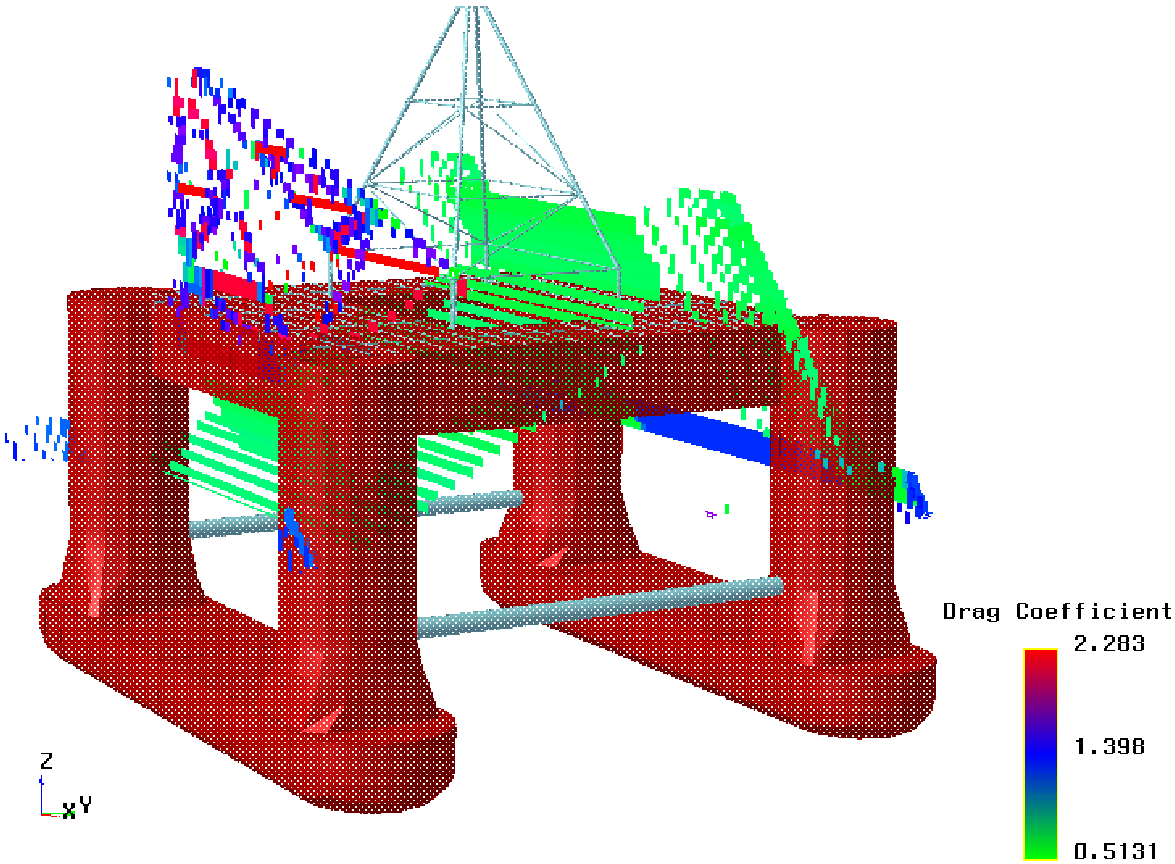
5.4.1.8 Graphical Results

HydroD may present animations of the heel motion, both with the deck compartments included and without.

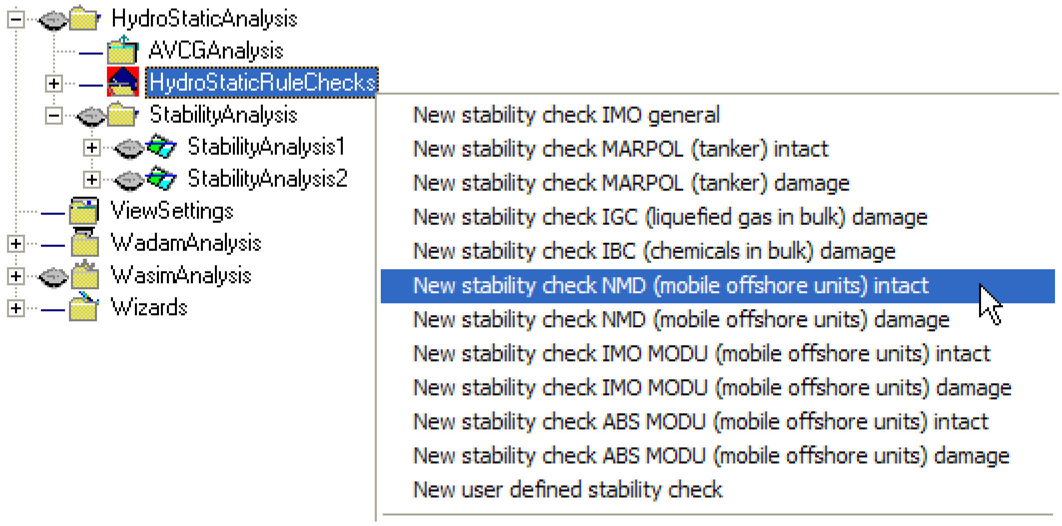




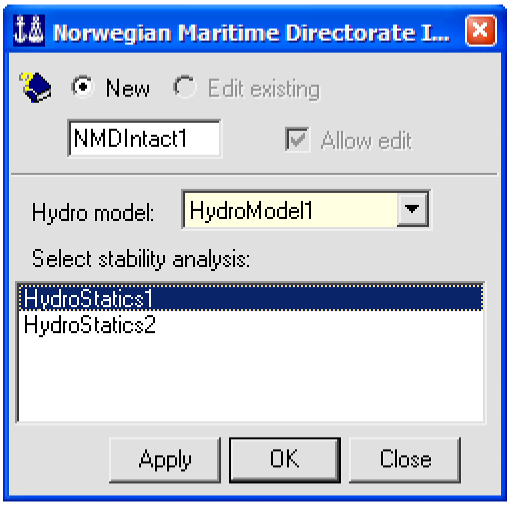
The results of the wind calculation may also be displayed, in form of colour plot of drag coefficients. By selecting the ‘Modelling Draw Style’ feature, the drag coefficients may be displayed for various heel angles (below shown for 30°).

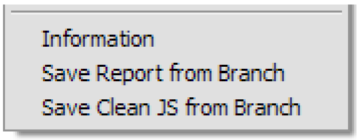


5.4.1.9 Hydrostatic Rule Checks

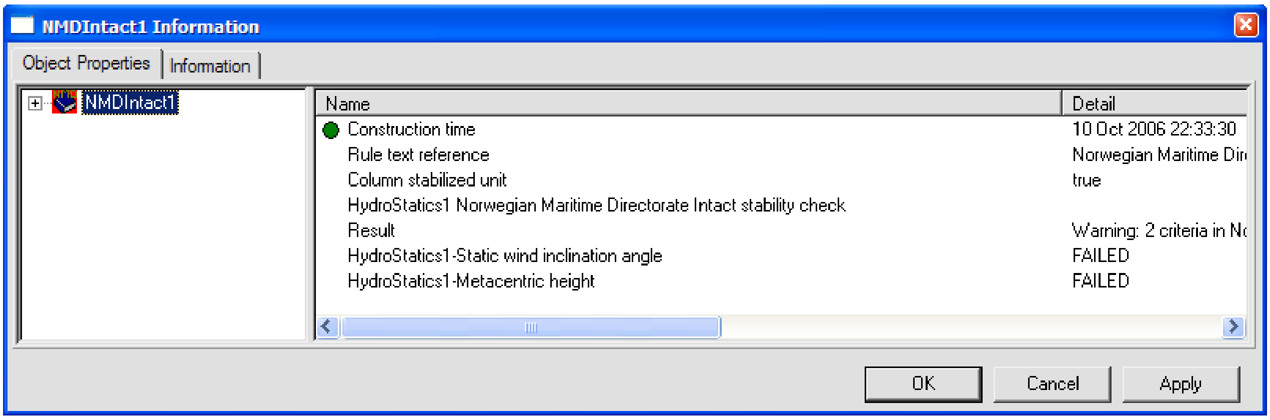


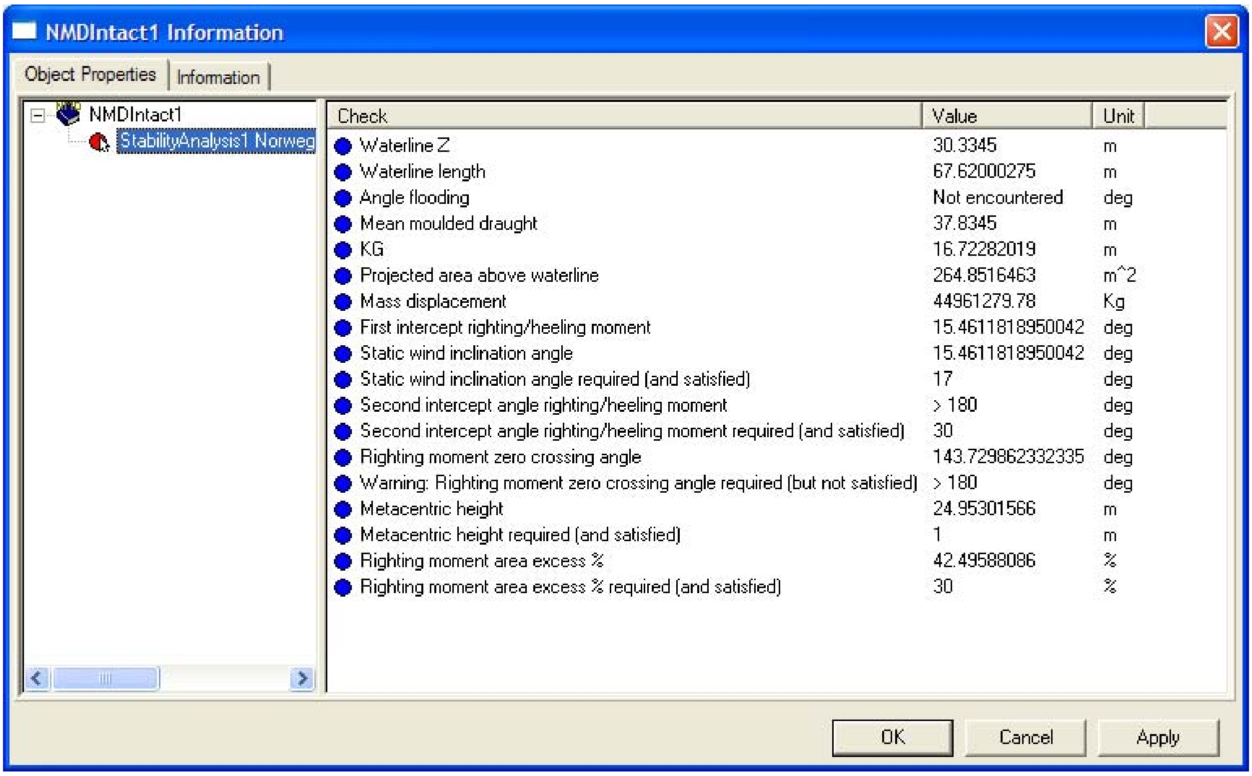
When the hydrostatic analysis has been performed, hydrostatic rule checks may be carried out based on various given codes.





The results of this code check may be inspected by use of the Information option, given below, or as a saved report.



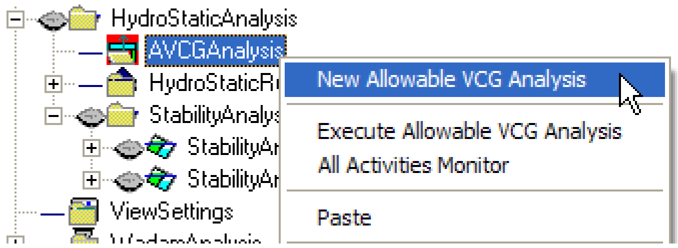


5.4.1.10 Damage Stability Analysis

In order to perform stability analysis in damaged condition, the following approach is recommended:

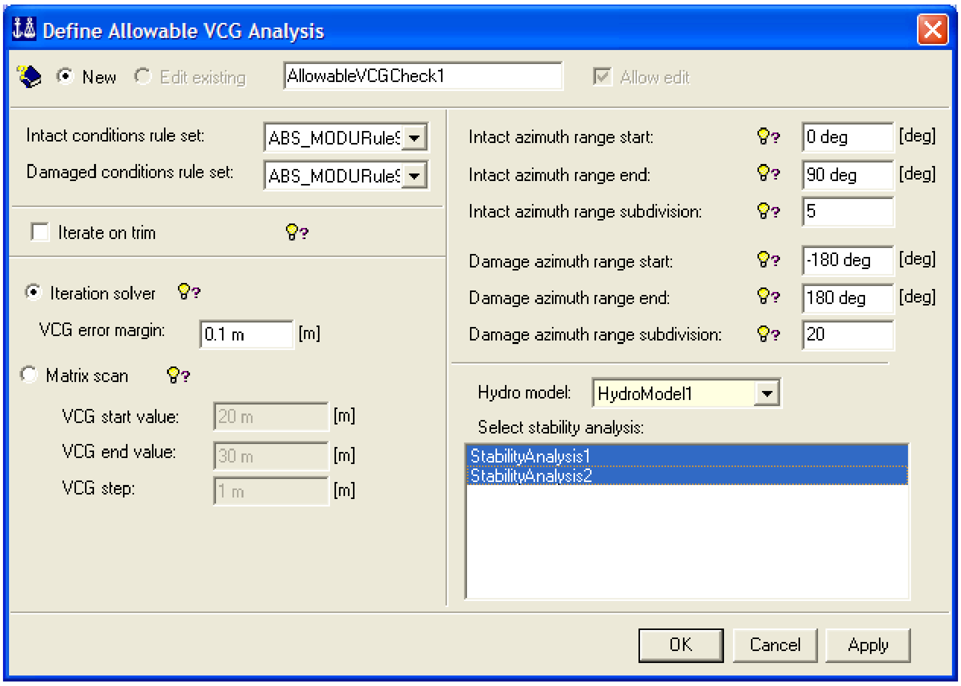
- Establish an intact loading condition equal to the damaged condition prior to the relevant compartments getting damaged.
- Establish the damaged condition by copying the intact condition and applying a flooded property to the damaged compartments. Re-compute the equilibrium position (that will now typically be with a heel angle). It is the equilibrium position that will be the outset of the stability analysis (and hydrodynamic analysis on the same loading condition if desired).
- Establish a stability analysis that uses the damaged condition as its loading condition and the intact version of the damaged condition as its undamaged condition.
- Use the “Auto compute rotation axis” button to find the correct rotation axis.

5.4.1.11 AVCG (Allowable Vertical Centre of Gravity) analysis



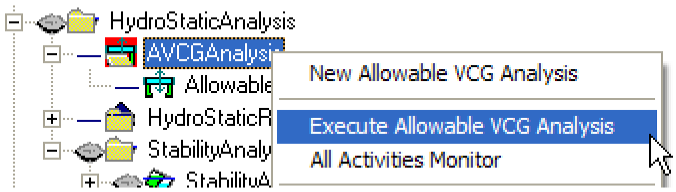
An AVCG analysis is an analysis where the maximum allowable vertical position of the centre of gravity according to a specified rule set, is computed for a (normally large) number of intact and damaged loading conditions. There are several Rule sets to use in these analyses.

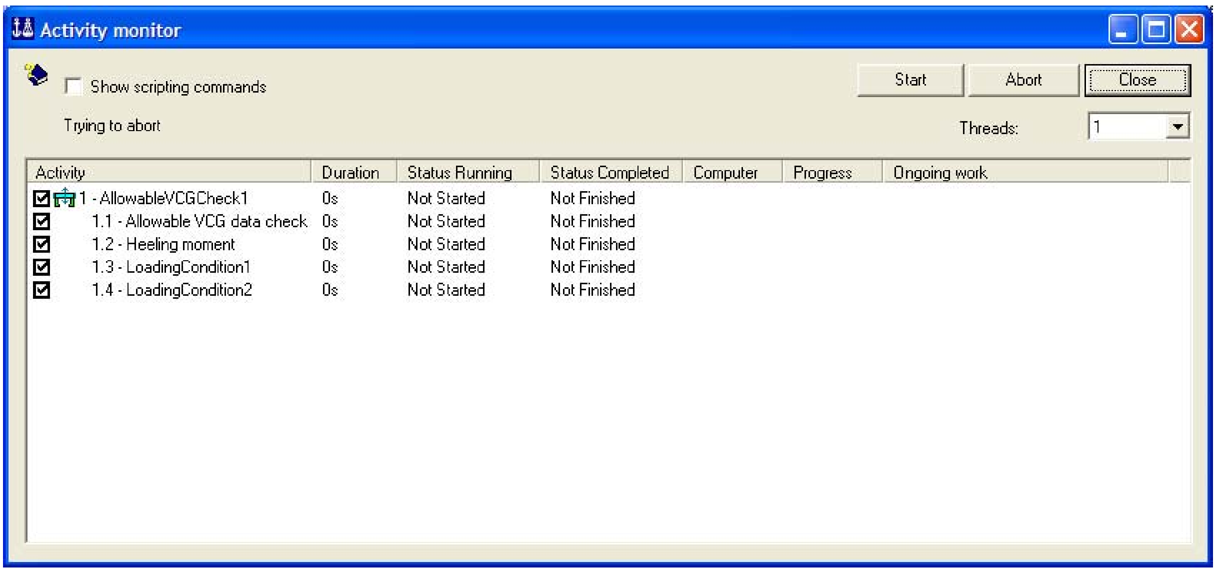
A part of the analysis is to find the most critical rotation axis. The range of axis directions to search through is defined by the user. The menu for defining and AVCG analysis looks as follows:



Normally the list of stability analyses is a lot longer than in this example. Notice that this type of analysis can take a considerable amount of CPU time since a wide range of loading conditions must be analysed for many different rotation angles. The light bulbs in the dialogue contain detailed description about the different input parameters in the analysis.

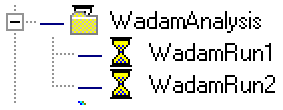
The AVCG analysis is executed as follows:





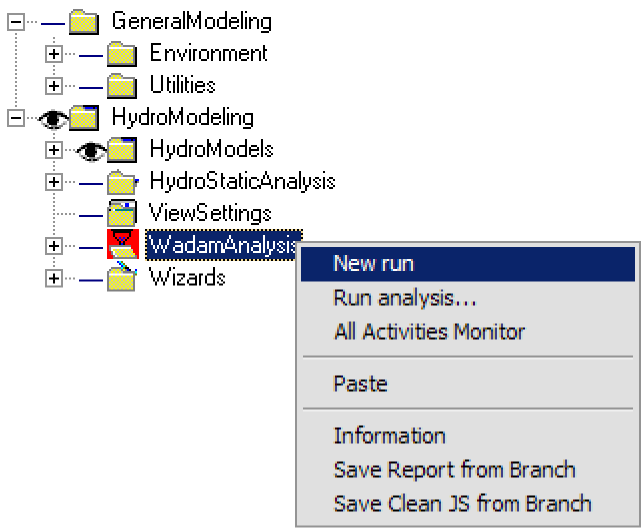
5.4.2 Hydrodynamic Analysis by Wadam

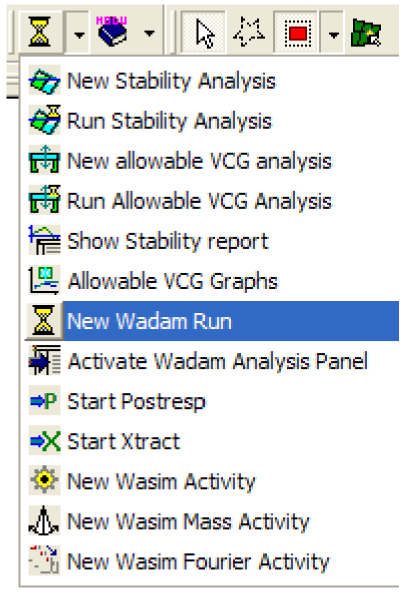
The Wadam Analysis folder contains all the Wadam runs.



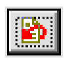
5.4.2.1 Wadam Run

A new run may be defined by right clicking the browser, or by use of the toolbar.



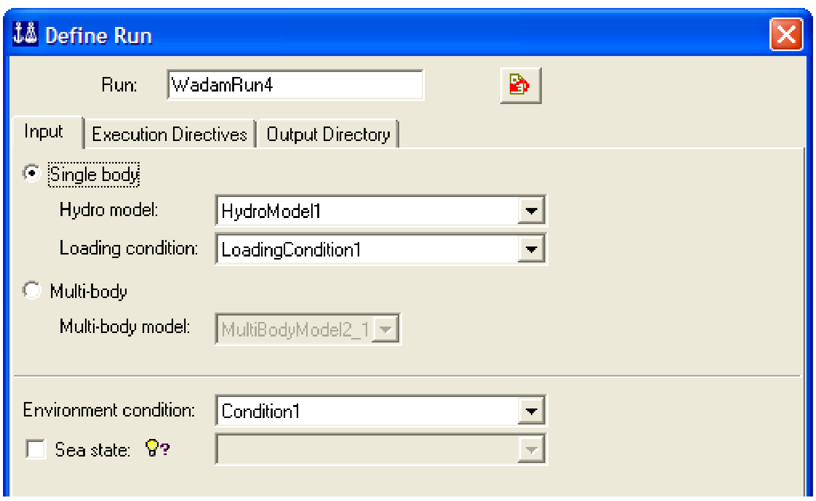


A Wadam run contains all the information required to execute Wadam. It holds environment data, a hydro model and a number of execution directives, defined in different tabs in the dialog window. All execution directives are also documented by hints in the dialog.

By clicking the toolbar button, the information in this dialog will be reset to default settings. 

The dialog window contains different tabs, as explained below.

The Input tab



The analysis will be either a Single body analysis or a Multi-body analysis.

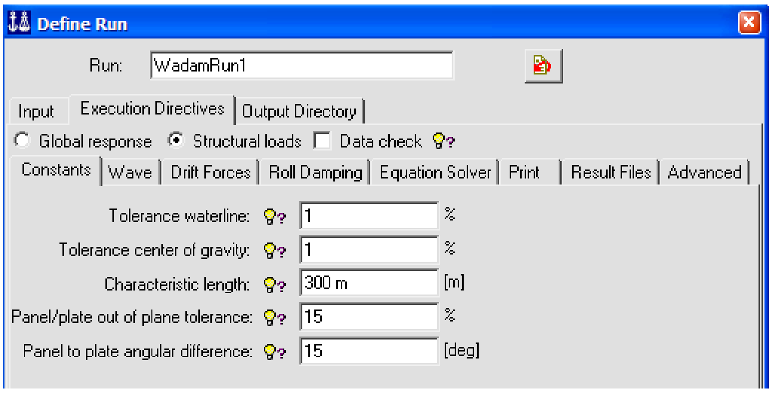
- Hydro model — Select the correct hydro model (there may be several models in the database)
- Loading condition — Select the correct loading condition (which contains e.g. the mass model).
- Multi-body — Select the correct multi-body model
- Environment condition — Select the correct set of directions and frequencies/periods
- Sea state — Select the correct sea state. To be used for roll damping calculation and drag linearization.

The Execution Directives tab

Analysis Type

- Global response — A global response analysis will extract motion characteristics, global forces, wave kinematics etc.
- Structural loads — The structural loads will create a detailed load calculation for a structural model, in addition to the global response analysis
- Data check — A data check will go through the input data to determine whether they are consistent. You have to go into the Wadam.lis file to inspect the results. Wadam will terminate after the data check is performed.

Constants



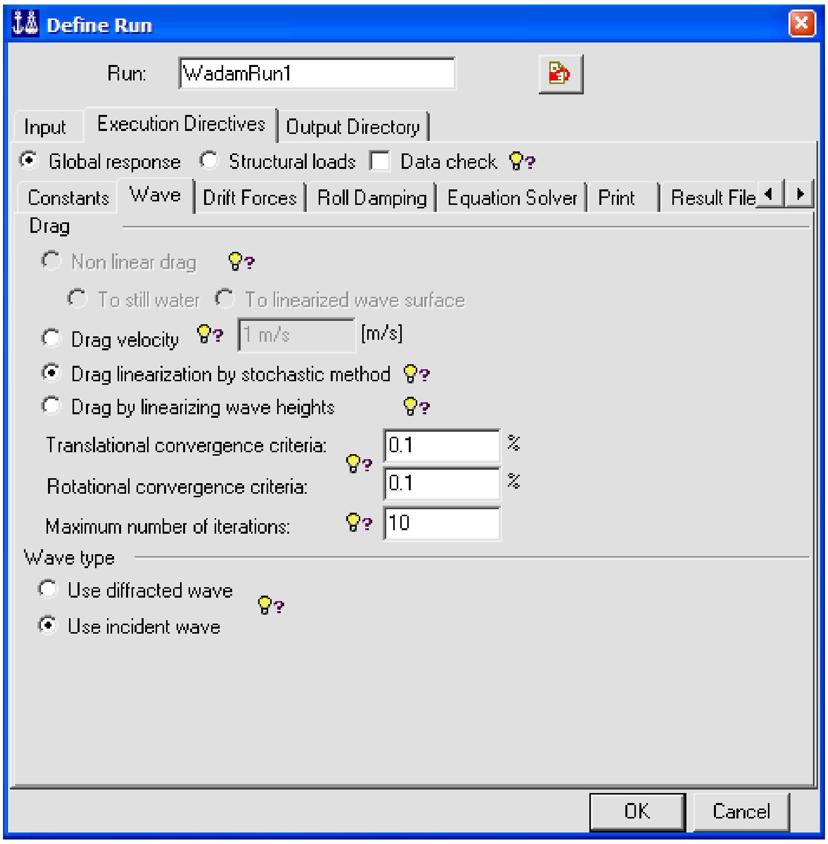
- Tolerance waterline — Tolerance on the computed z-coordinate of the waterline as a percentage of the characteristic length.
- Tolerance centre of gravity — Tolerance on the horizontal distance between centre of gravity and centre of buoyancy as a percentage of the characteristic length
- Characteristic Length — Usually the largest horizontal distance between 2 points on the average immerged surface
- Panel/plate out of plane tolerance — Maximum difference between panel area and panel to structural plate pyramid surface area. Value is given in percent.
- Panel to plate angular difference — Maximum angular difference between the panel surface normal and the structural model plate surface normal.

Notice that Wadam will stop if the first two tolerances are exceeded. These are controlling the equilibrium between the buoyancy and mass of the model.

The last two tolerances are used for mapping of the dynamic water pressure from the panel model to the structural model in a load transfer analysis.

Wave

Specify how to handle computation of drag forces from a Morison model.



Drag

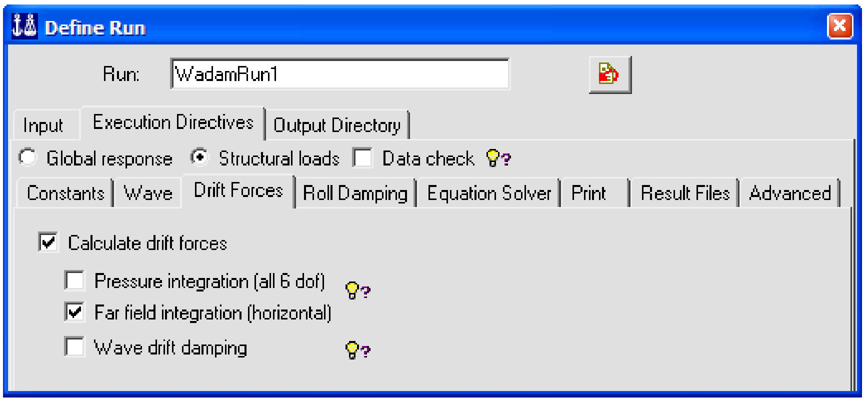
- Non-linear drag — Option for time domain (deterministic) output only
- Drag velocity — Specify linearization velocity to be used on all Morison elements to calculate the drag force. The same velocity is used for all headings and frequencies.
- Stochastic method — Estimate linearization matrix based on the most probable largest motion response assuming a Rayleigh distribution. Both forces and damping will be updated according to this matrix. (Requires that a wave spectrum is defined)
- Wave heights — Estimate linearization matrix based on the response given by a wave height surface. Both forces and damping will be updated according to this matrix.
- Convergence criteria — Give the convergence criterion for the translational and angular modes in percentage of the consecutive error (no more iterations are performed when two consecutive runs differ by less than the convergence criteria).
- Iterations — Even if the convergence criteria have not been reached, no more iteration will be performed when max number of iterations has been reached.

Wave type

- Calculate Morison forces based on the diffracted wave (wave field resulting from the potential theory calculation) or from the incident wave (the incoming wave field). Diffracted wave may only be used if the hydro model is fixed and a panel model is employed in the calculation.

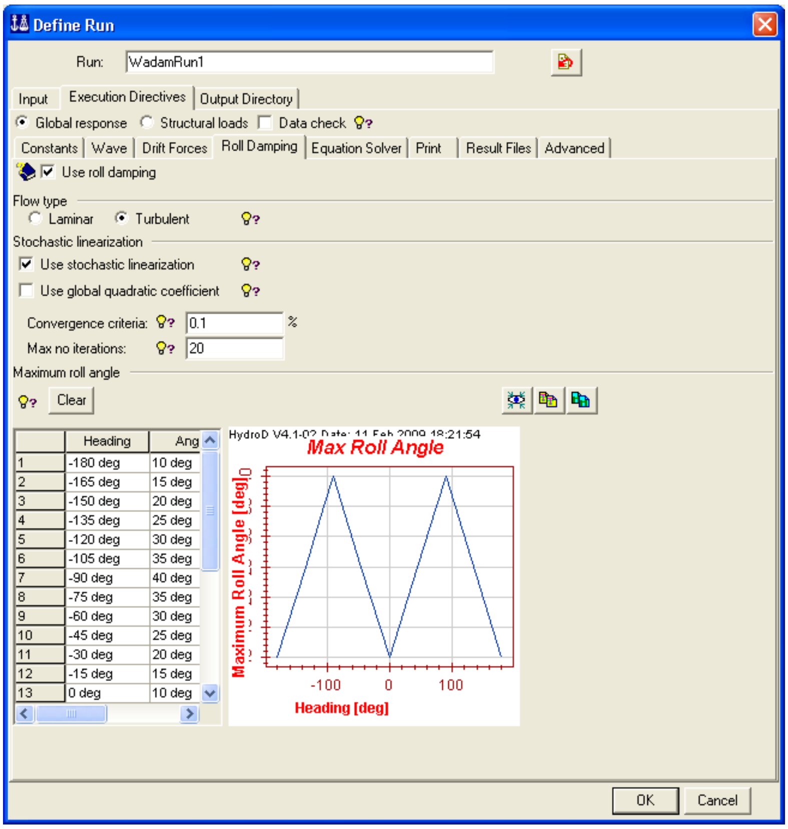
Drift Forces

Calculate second order mean drift forces.



- Pressure integration — Calculated by pressure integration in the six degrees of freedom
- Far field integration — Calculated by momentum conservation in the three horizontal degrees of freedom.
- Wave Drift Damping — Calculate the 3 by 3 wave drift damping matrix due to slow motion in the three horizontal modes surge, sway and yaw.

Roll Damping



Calculate the resulting roll damping by definition of a separate strip model. As the roll angle is both an input parameter and a part of the solution, this calculation has to be an iteration process.

For stochastic linearization a quadratic coefficient can also be used. This must be defined in a "Loading Condition – Additional matrices – Roll Damping – Non linear coefficient" property. When using the non linear coefficient, the maximum roll angle will not be available.

The iteration may be performed automatically by Wadam for a maximum roll angle from short term statistics. This requires the definition of a wave spectre in the relevant sea state. More details on this calculation are found in the user manual for Wadam.

Input:

- Flow type — Define laminar or turbulent (recommended) flow around the ship hull.
- Stochastic iteration — Turn the automatic iteration on.
- Use global quadratic coefficient — Turn the quadratic coefficient on.
- Convergence criteria — Define criteria in percent of consecutive error.
- Max no iterations — Specify the maximum number of iterations attempted to meet the convergence criteria
- Maximum roll angle — Specify pairs of heading and maximum estimated roll angle

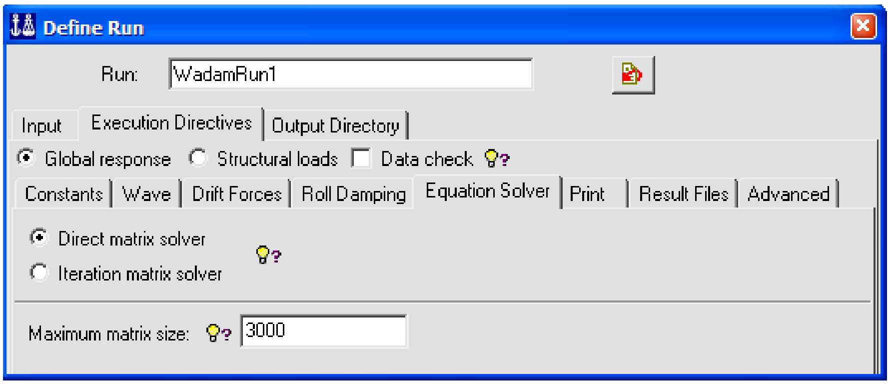
If stochastic roll angle iteration is performed, the maximum roll angle values are employed as start value in the iterations. If the headings do not correspond to the input headings given under environment, linear interpolation between the two closest headings will be used.

The curve is used to calculate a linearized viscous and eddy-making damping and will therefore significantly affect the transfer functions at wavelengths close to resonance.

Notice that the angles must be larger than 0. By specifying a very small angle (0.001) all viscous effects will be neglected.

Equation Solver

Specify the type of solver for the linear system of equations for the velocity potentials.

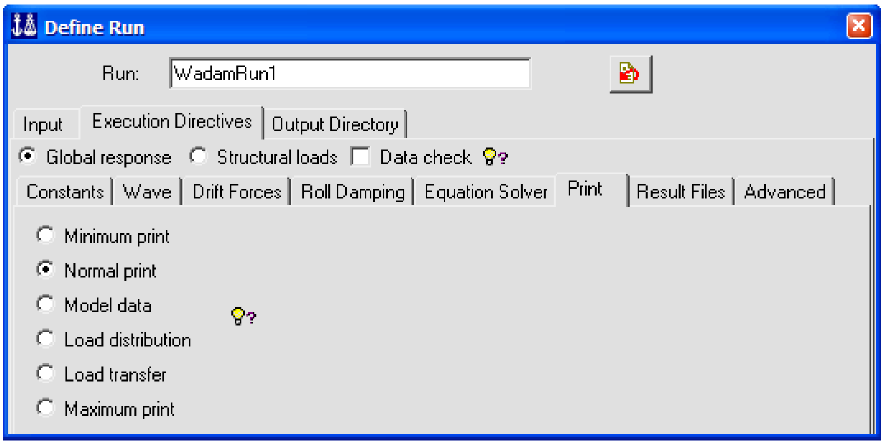


- Direct matrix solver — The equation system is solved by a direct method. This is the recommended option when number of panels is less than 3000 panels (depending on the amount of memory available. Direct method must be used with difficult waterline shapes and thin bodies.
- Iteration matrix solver — The equation system is solved by iteration. The iteration solver may sometimes fail due to special geometry or symmetry conditions.

- Maximum matrix size — If the model contains more panels than the maximum size, a block iterative solver will be used.

Print

This setting controls the amount of information printed to the listing file, wadam1.lis.

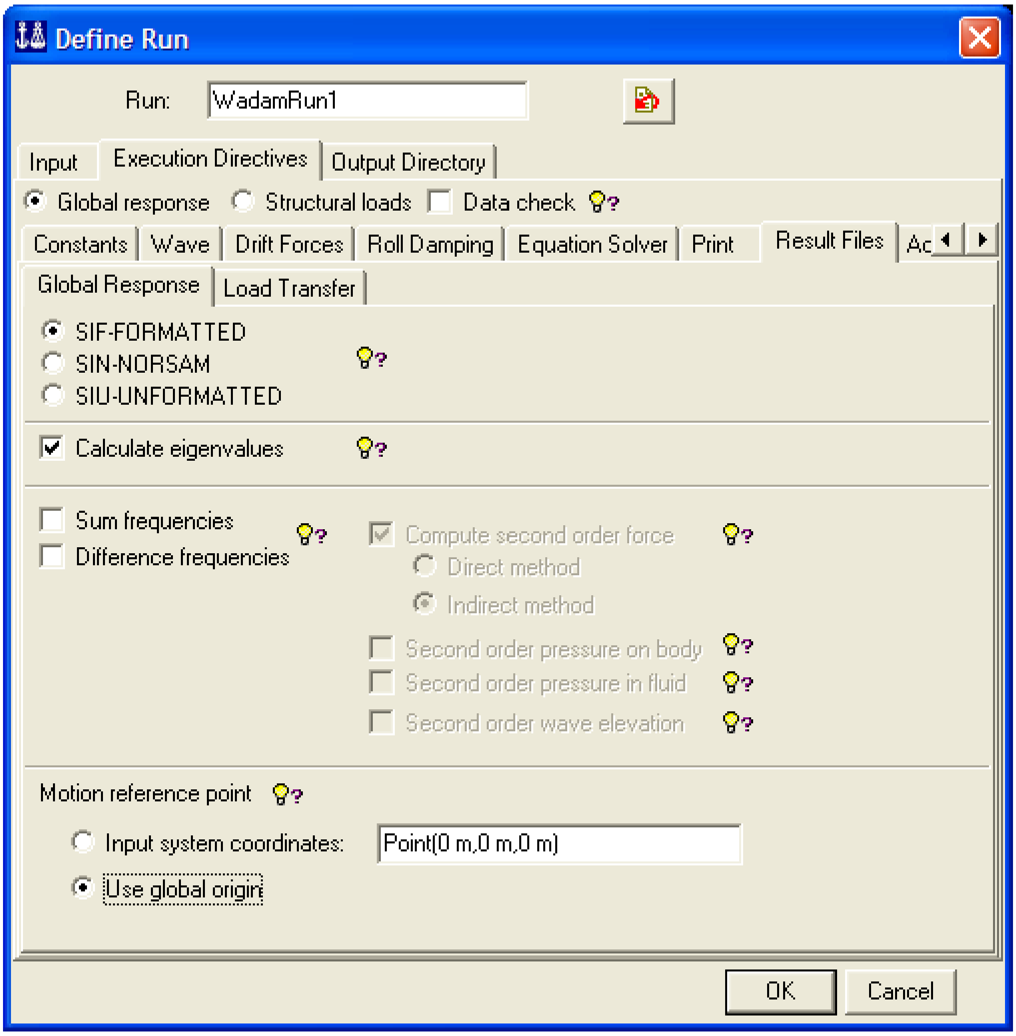


- Minimum amount of print provides only the most important information.
- Normal print provides the most relevant printing information (default).
- Print model data gives detailed model data in addition to normal print.
- Print load distribution gives detailed load distribution in addition to what print model gives.
- Print load transfer gives detailed loading on all elements in the Morison model in addition to print load distribution.
- Maximum print provides an extensive amount of maintenance print. It is usually only suitable for debugging purposes.

Result Files

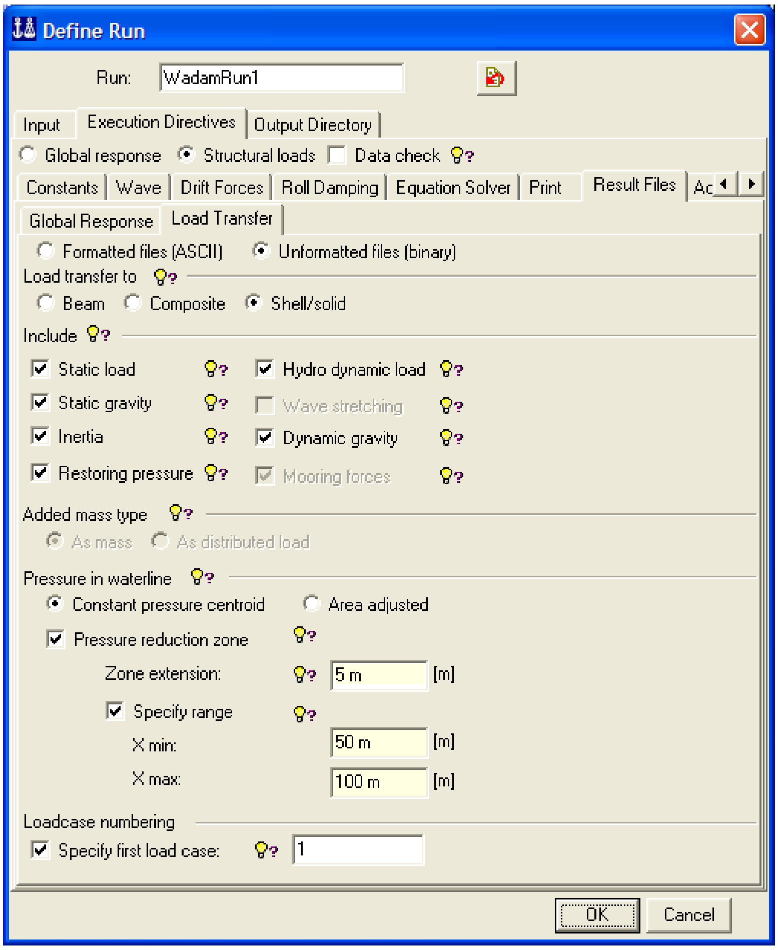
Different types of results files may be selected, depending on the analysis type. The different files are described in the section Output Files.

Global Response



• SIF/SIN/SIU	Generate global response result files to be post processed in applications like Postresp and Xtract.
• Calculate Eigenvalues	Calculate natural periods (eigenvalues) for the rigid body motion degrees of freedom. Eigenvalues are only relevant for floating bodies.
• Sum and Difference frequencies	Relevant for second order results. This option is explained in more detail in the Wadam user manual.
• Reference point	The global response reference point may be defined here. The coordinates can be specified in the **input** coordinate system or at the global coordinate system origin.

Load Transfer



File formats

- Formatted files   Load files in text format
- Unformatted files   Binary load files (recommended)

Structural Model types

- Beam model   All loads are transferred to a structural beam model. The Morison model must be one of the super elements in this model.
- Composite model   Morison loads are transferred to the beam part of the structural model and Panel loads are transferred to the shell part of the structural model. One of the super elements in this model must contain the same beam elements as the Morison model (or simply be the same model).
  Alternatively, a single superelement composite model may be used, where the model is a complete first level superelement model, containing both the Morison parts and shell elements. In such a case, the Morison model and the structural model must be identified by the same superelement.
- Shell/solid model   Panel loads are transferred to the shell/solid elements of the structure model.

Load types

- Static load   Include hydrostatic load in the load file

- Static gravity — Include gravity forces in the static load case (default on when static load is included)
- Inertia — Include inertia forces in the load file (default on)
- Restoring pressure — Include restoring pressure forces on the loads files. (Fluctuating hydrostatic pressure from the motion of the vessel) (default on)
- Hydrodynamic load — Include hydrodynamic load cases in the load file (default on)
- Wave stretching — Switch ON or OFF calculation of wave pressures to finite water surface by stretching when transferring pressures to the Loads Interface Files. This option is valid only for time domain.
- Dynamic gravity — Include fluctuating gravity in the dynamic load cases (default on)
- Mooring forces — Include anchor forces in the static and dynamic load files (default on, requires a Morison model)
- Added mass — For beam structural models you have the option of whether added mass should be included as mass or distributed beam load.

Pressure in Waterline

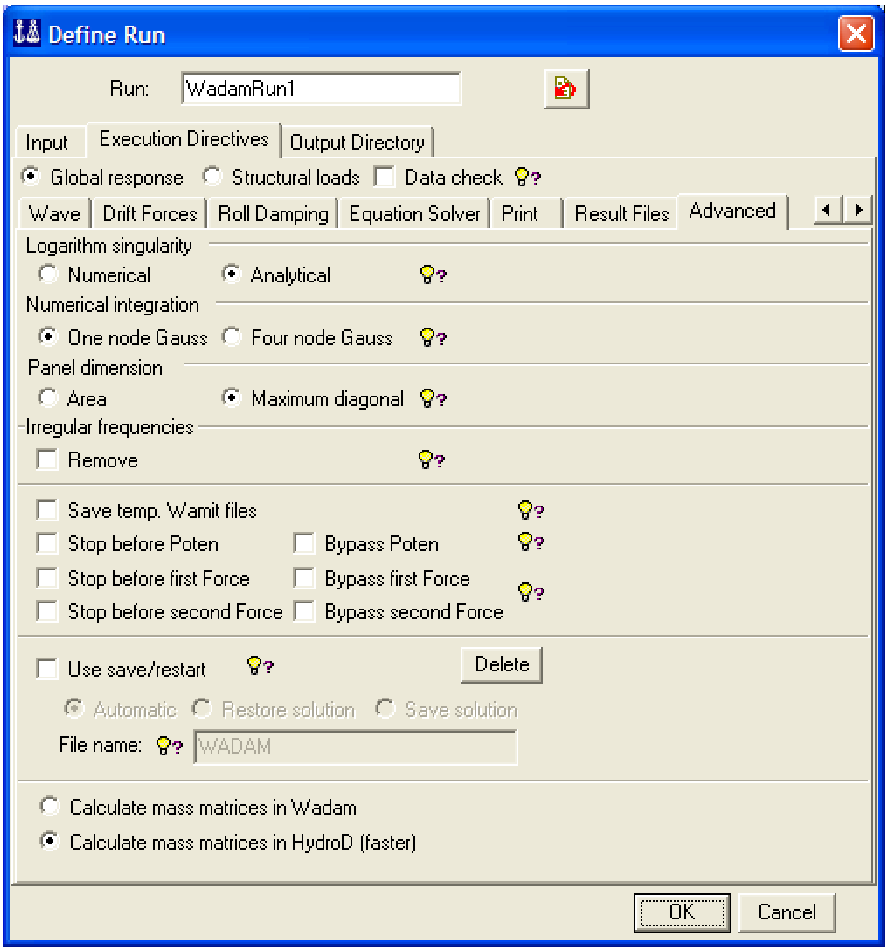
- Constant pressure centroid — The panel will receive pressure equal to the pressure at the centroid if the panel centroid is submerged.
- Area adjusted — The panel pressure is adjusted according to the submerged area
- Pressure reduction zone — The pressure will be modified in a zone around the free surface. Panels above the free surface will get a pressure load, and the pressure on panels below the free surface within the specified zone will be reduced.

  If “Specify range” is checked, the pressure reduction may be defined in a range on the x-axis. The constant pressure centroid or area adjusted method selected above will then be used outside this range.

Load case numbering

- Specify first Load Case Specify the lowest load case number used by Wadam when generating loads files (we recommend to start with load case 1). The S#.FEM file is only created when the load case numbering starts at 1. The S-file is required for any subsequent stochastic fatigue analysis.
- The load cases for a Morison model will always start with load case 1.

Advanced



Most of these settings are normally left as default values. They may be used for special types of analyses. The top three options regard different methods for solving the radiation-diffraction problem.

Removal of irregular frequencies may be done in special cases.

The part with Save and Stop options are used for special investigations.

For all of these, we recommend studying the tool tips in the dialog window in addition to the descriptions in the user manual for Wadam

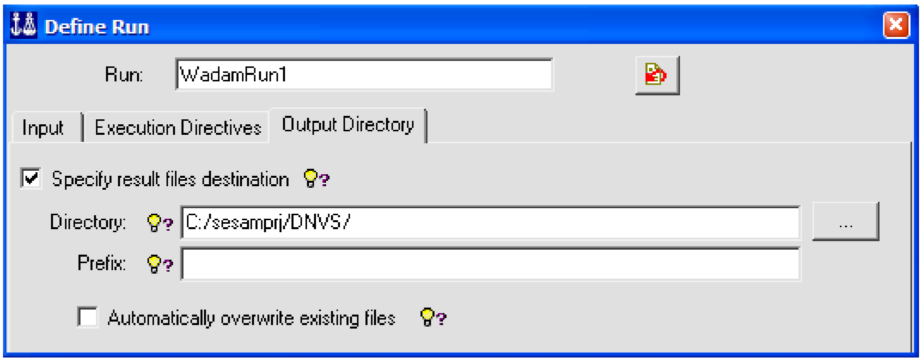
The bottom option, Save/Restart, is a more frequently used feature. This may be used to reuse the radiation/diffraction potential solution, instead of calculating this once more when only other parameters are changed in a new run.

- Automatic — Wadam will reuse existing potentials and append (generate new file if necessary) non-existing potentials to the save file.
- Restore solution — Wadam will reuse existing potential solutions where these can be found.
- Save solution — Wadam will save the calculated potentials on a new file.
- File name — Each run must specify the name of its save/restart file. Different runs may reuse the same save restart file. Runs having different potential solutions must have different names on the save/restart file. If you for instance want to execute two runs that contain different loading conditions, the second run may NOT reuse the same save/restart file name.

Note that the reuse of a save/restart file is dependent upon no change in the Panel model or other data that may change the potential solution. Wadam does no automatic checking on whether the potential solution actually can be reused.

Press the delete button to remove all existing restart files (relevant if you have made changes that invalidate the restart files).

Output Directory tab



By default all result files are put in workspace directory as defined by the run name:

`…/Workspace_path/run_name/result_file`

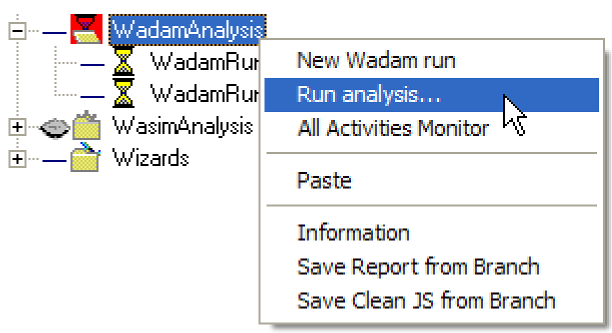
Normally this is the best choice.

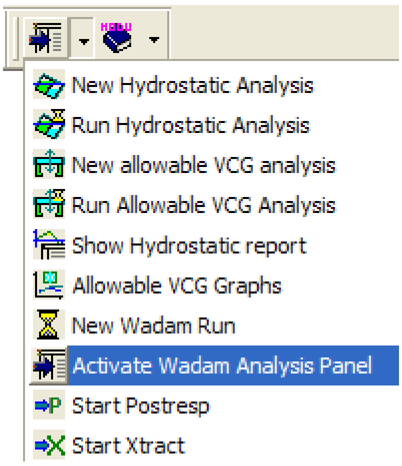
If however a subsequent structural analysis is to be run in a different directory, this can be specified here. By giving a directory name and a file prefix you can direct the result files to a destination of your choice.

The G*.SIF/SIN/SIU, S*.FEM and L*.FEM files will be moved to the directory you specify.

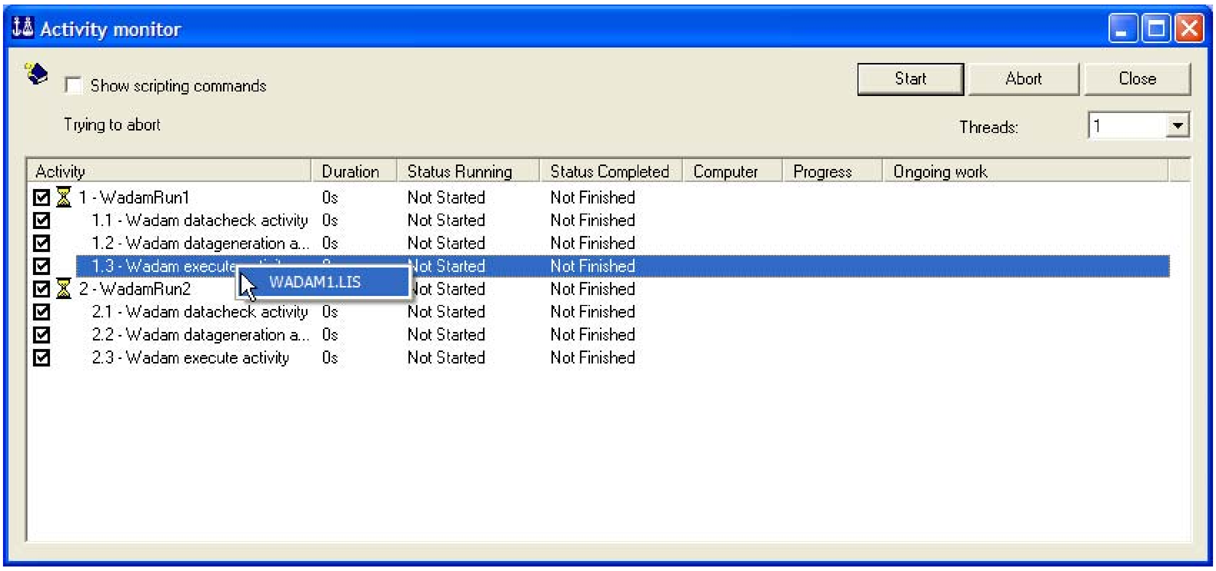
5.4.2.2 Execute Wadam analysis

Once the run objects have been defined, you may go into the analysis panel. This can be accessed by right clicking the WadamAnalysis folder. Select either Run Analysis or All Activities Monitor:





Alternatively use the toolbar button to activate the Analysis Panel.

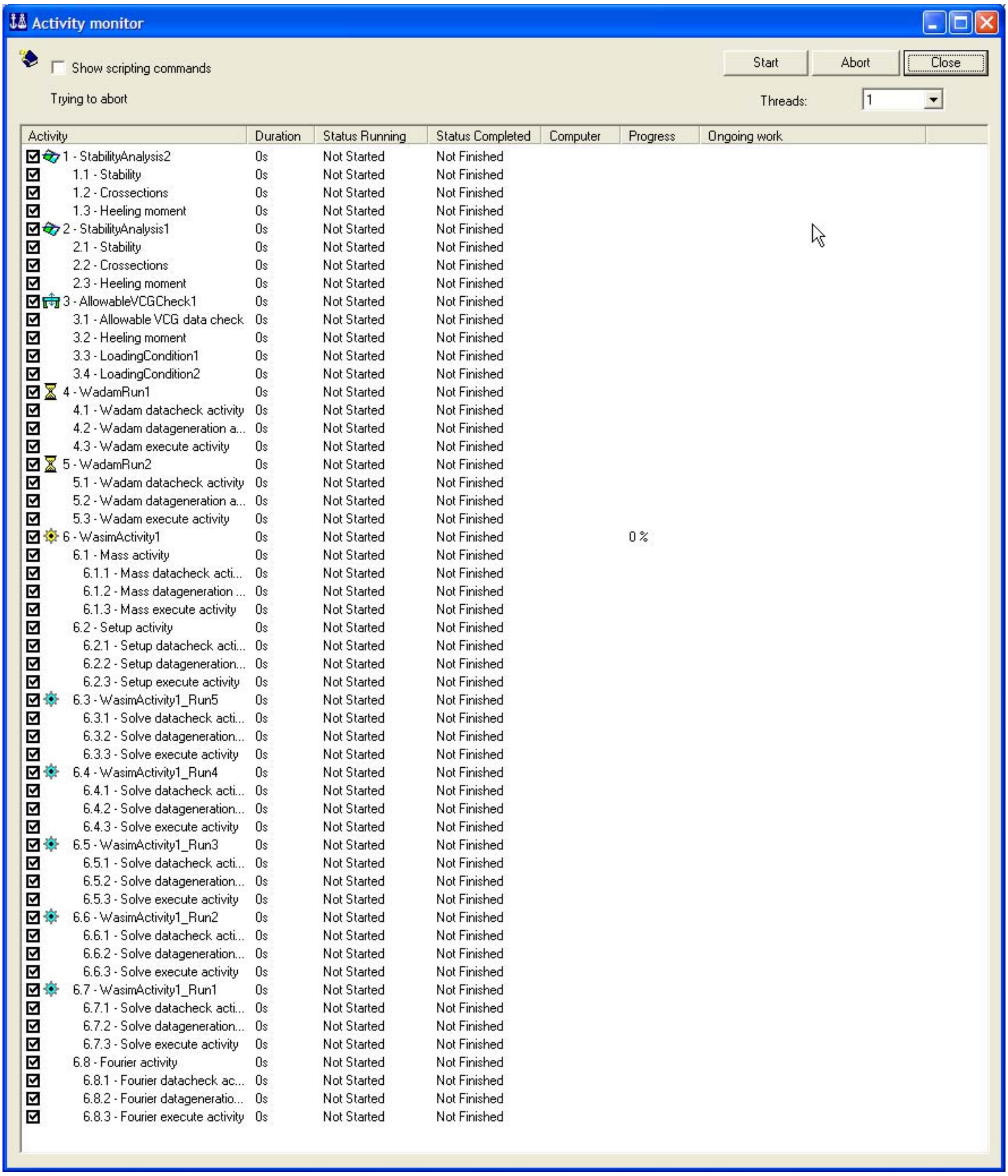


In the Analysis Panel you tick for those activities that you want to execute. By default all activities are selected. Press "Start" to execute all selected activities.

After completion of a Wadam execute activity you can inspect the print (LIS) file by selecting the activity, and then select Wadam1.LIS from the RMB list.

Alternatively you can also select the All Activities Monitor. This includes any defined hydrostatic and hydrodynamic analyses, and may looks as shown below. Again you can select all the activities you want to execute and start the sequence by using the "Start" button. By default all activities are selected. You can also open print files in the same way as shown above. To journal the execute commands on the js-file you need to tick the "Show Scripting commands" box.

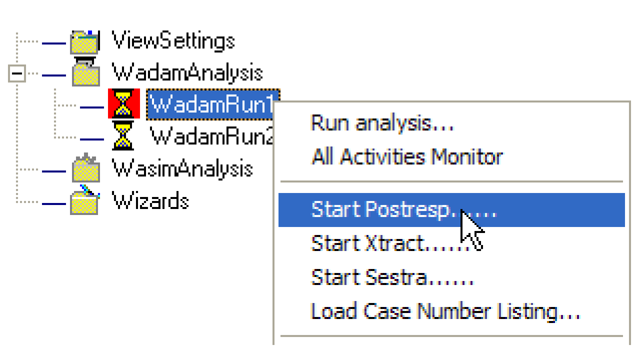
On a PC with several processors it is possible to execute several activities in parallel. The number of parallel activities is specified in the "Threads" field.

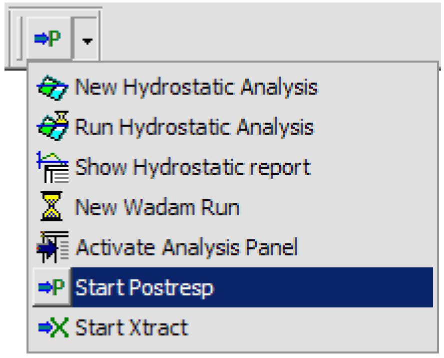


5.4.2.3 Hydrodynamic Results

The listing file from Wadam may be accessed directly from HydroD, as described above. This file contains results from the analysis, such as hydrostatic equilibrium, sectional loads and motions. This file should always be inspected, also for any possible messages from Wadam.

Once a run is completed, you can start Postresp, Sestra and Xtract from the browser or from the toolbar button:





You must then first select the relevant run in the browser.

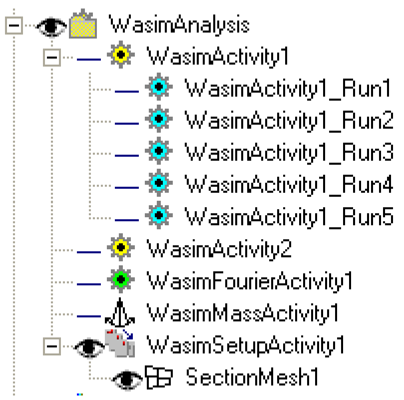
Postresp may be used to display and perform computations on results like transfer functions, including statistical calculations.

Sestra may also be started from this menu. HydroD will generate a Sestra input file using the structure model as the top level element, and execute Sestra.

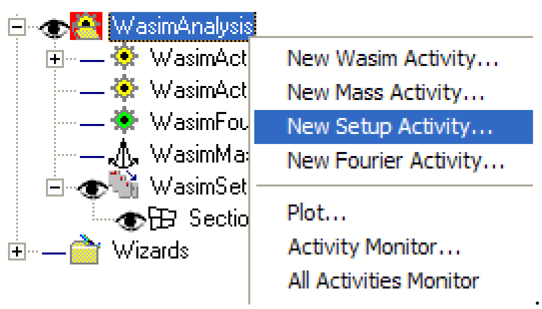
Xtract may be used to display the model, including the calculated wave load pressures, and to make animations of the motion of the structure.

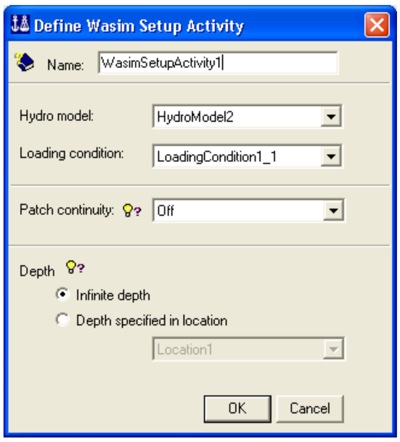
5.4.3 Hydrodynamic analysis by Wasim

The Wasim analysis folder contains al the different types of Wasim activities.



5.4.3.1 Setup activity

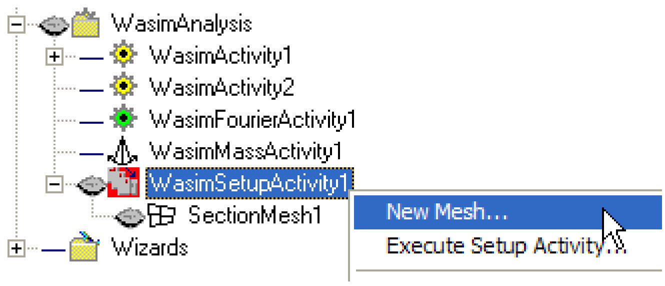




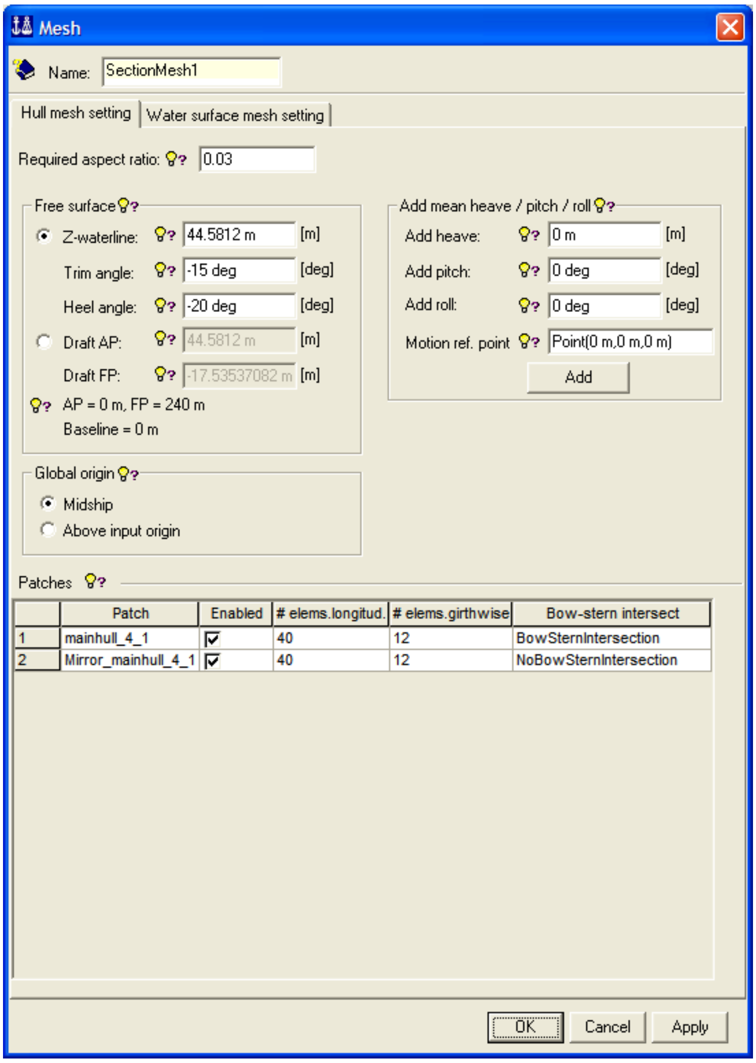
......

If both a loading condition and a specified mesh is to be used in several Wasim analyses it is convenient to define a Setup activity. A setup activity refers to a loading condition (which specifies the waterline). The activity also contains a mesh and computes data that are given from the mesh only. Both the mesh and these data may then be reused in a subsequent Wasim activity.

The mesh is created from the RMB menu on the Setup activity:



The mesh dialogue looks as shown below. The generation of a SectionHullMesh on a Section model uses exactly the same dialogue, except that the “Water surface mesh setting” tab is not present in that case.

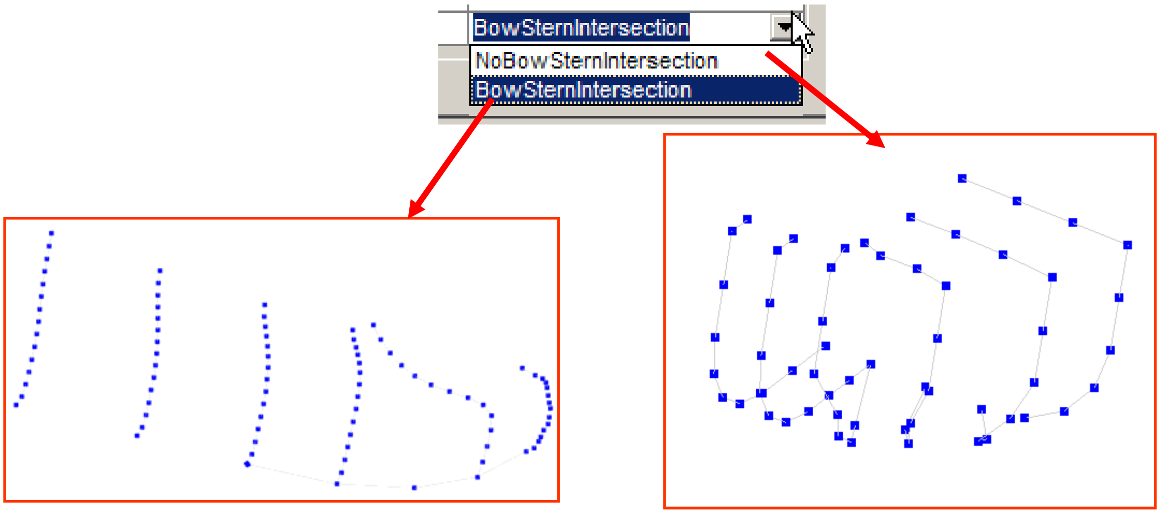


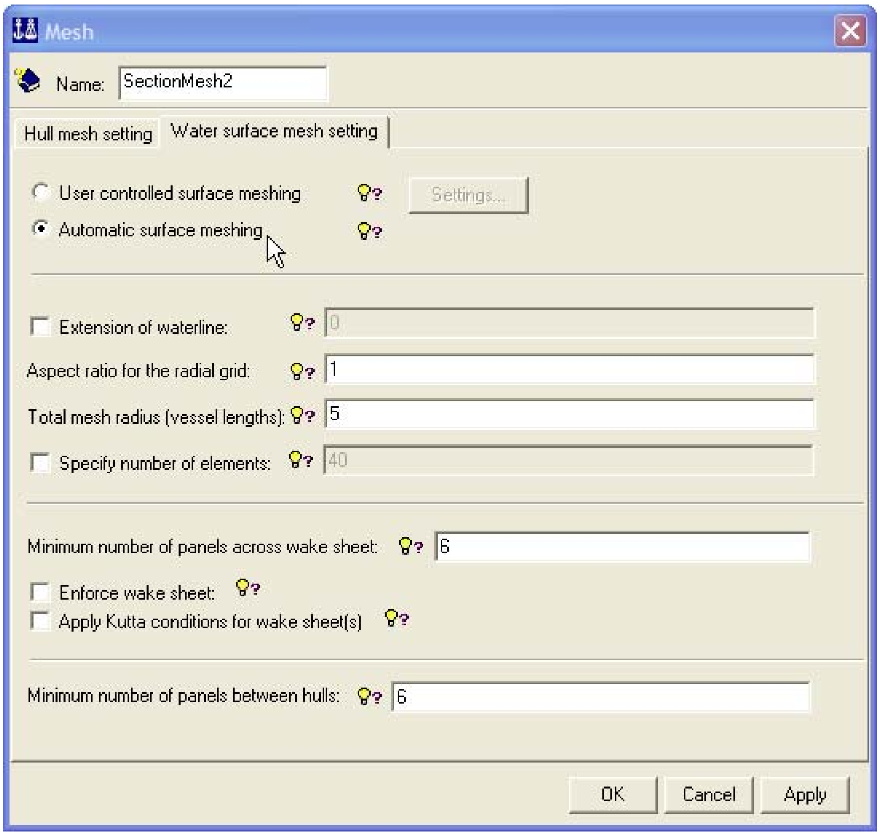
Hull mesh settings

The draft is by default taken from the specified loading condition, but may be modified. This could e.g. be to account for sinkage and trim due to forward speed. In such a case we recommend to use the “Add mean heave/pitch/roll” to find the modified draft values.

The data in the table at the bottom of the dialogue are by default set to the values specified for the SectionHullMesh (see Section model). Patches can be excluded from the mesh by removing the tick in the first column. The options in the last column are used in two different cases. The option “NoBowSternIntersection” is in general more robust and general. It can be used on cases like the patch shown right below. The limitation on the curves here is that no curve can intersect other curves. For ships it is in certain cases beneficial to allow the bow and stern curves to intersect other curves. An example of such a patch is shown left below. Then the option “BowSternIntersection” must be used. This option has

the additional limitation that the curves must be distinct in x, i.e. the smallest x-value on a curve must be larger than the largest x-value on the next curve. Thus the case right below would not mesh properly with this option. For additional information about the different fields see the light bulbs in the dialogue.





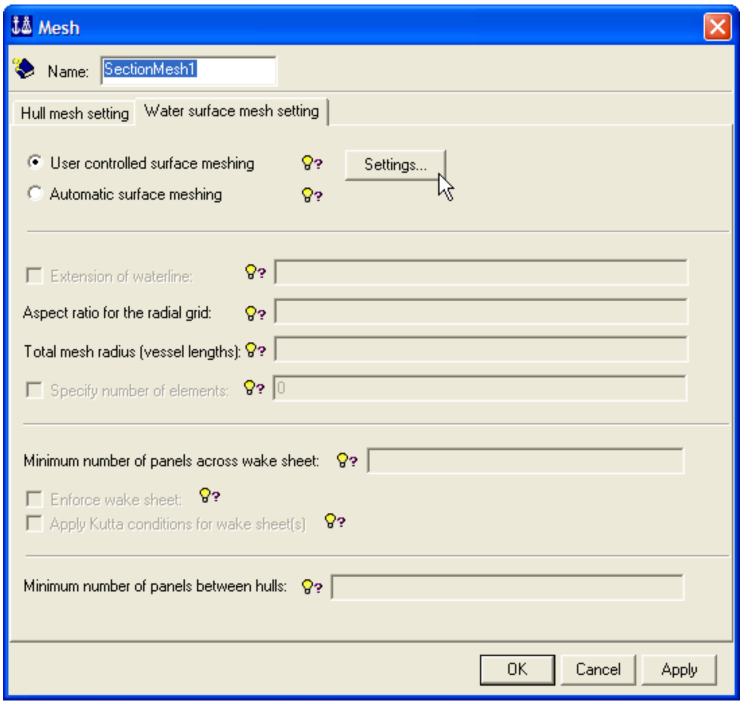
Water surface mesh setting

If the tick-box “Specify number of elements” is not checked the nodes on the free surface will match the hull nodes at the waterline. If it is checked a uniform mesh size with the specified number of elements is created along the waterline.

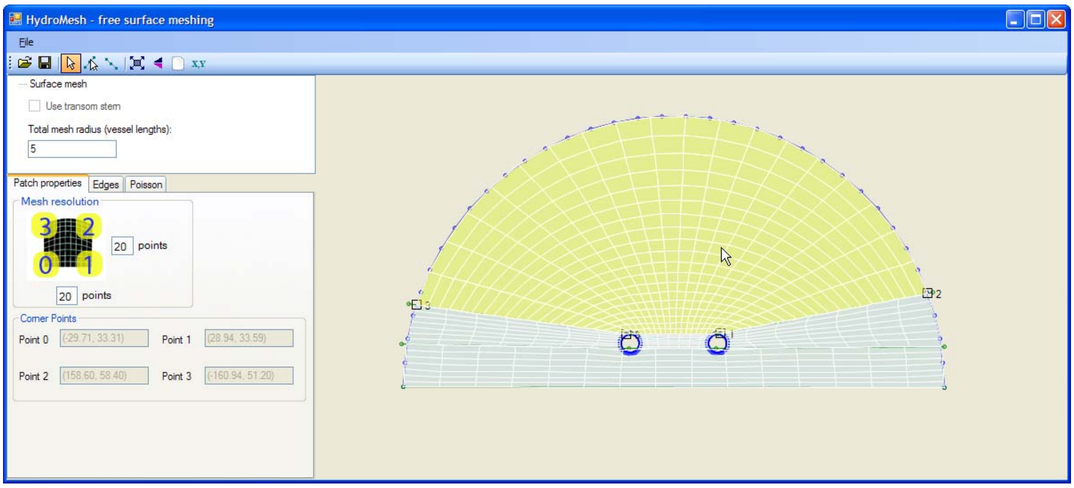
For the other input data on this tab, see the light bulb information for the different fields.

User controlled water surface meshing

This option is using the external program HydroMesh to create a user defined surface mesh. The user can define mesh "splitting lines" and control the resolution of the mesh patches. Select "User controlled meshing" and use the "Settings..." button to change the surface mesh.

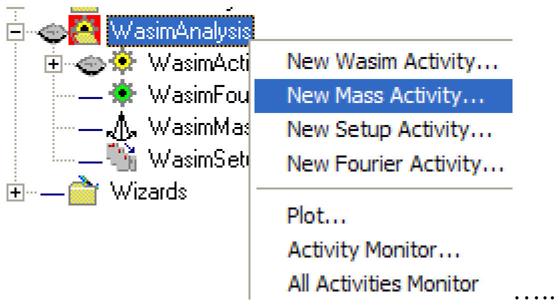


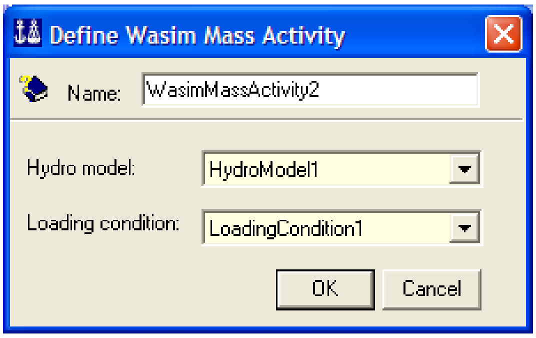
- Add surface mesh splitting lines. Click with the mouse to add points, press Enter to finish the line or Escape to cancel the change. The points coordinates can be edited later.
- Make sure the corner points (marked 0-4) are located in the right corners (select the patch and move them with the mouse)
- Set the mesh resolution for the patches.
- Define properties (point distribution) for the patch edges
- Save and close the window.
- See the HydroMesh user manual for more details about how to use this tool



5.4.3.2 Mass activity

If a loading condition and a set of load cross sections are to be used in several Wasim analyses it is convenient to define a Mass activity.

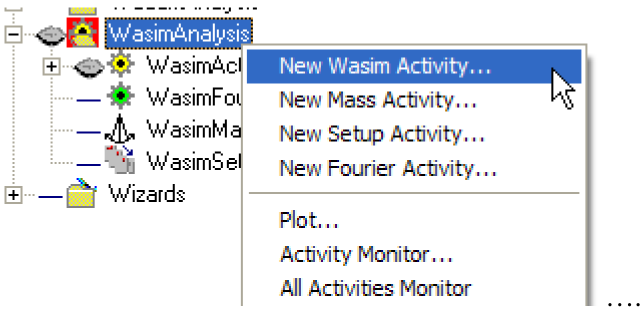


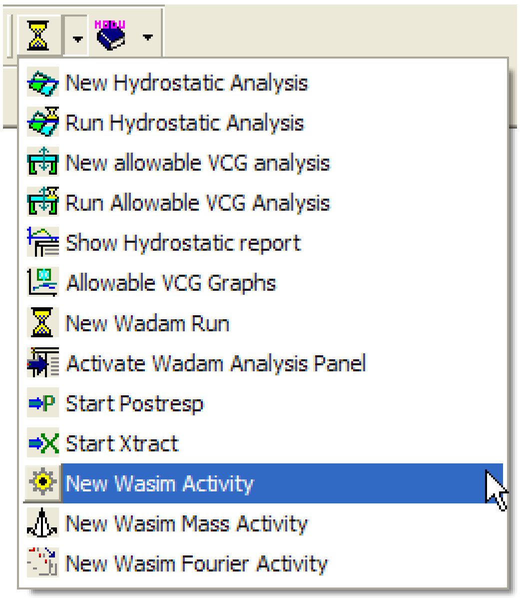


The output from the Mass activity is a file containing the global and sectional mass matrices for the structure. This file may be used in subsequent Wasim activities.

5.4.3.3 Wasim activity

A Wasim activity can contain Setup, Mass and Fourier as sub-activities, but can also reuse data from "external" Setup and Mass activities.





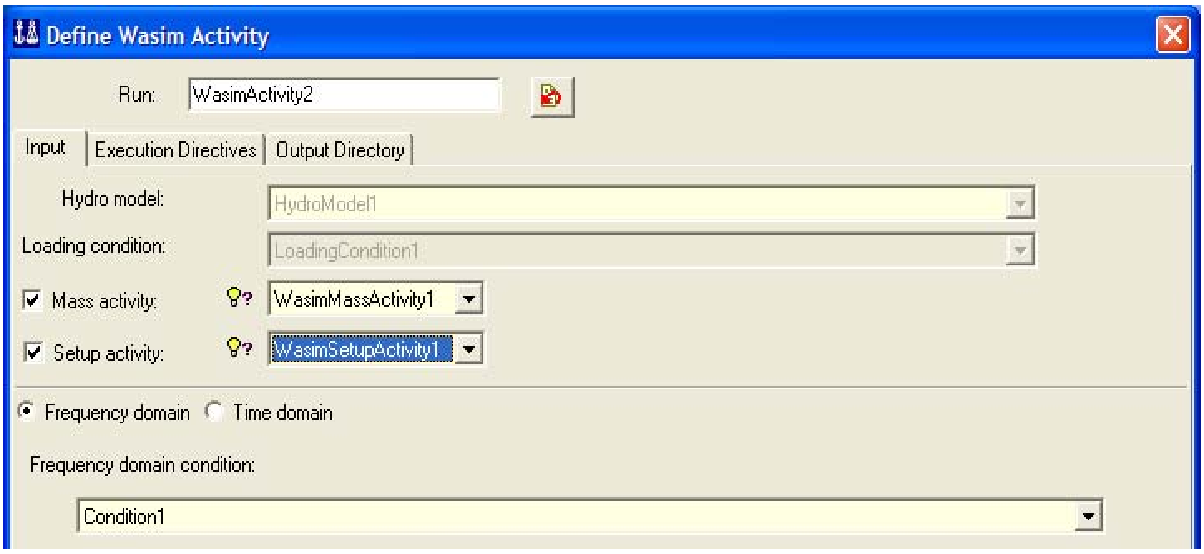
The Input tab

- Hydro model Select the correct hydro model (there may be several models in the database)
- Loading condition Select the correct loading condition (which contains e.g. the mass model).
- Mass Activity/Setup activity Select Mass/Setup activities to use in analysis.

If Mass and/or Setup activity is not checked, the activity will be included as a part of the Wasim activity.

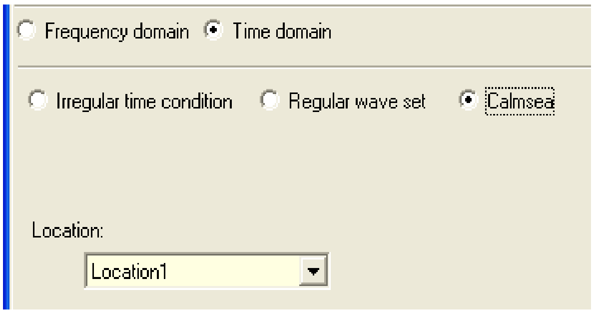
For frequency domain analysis reference is made to the frequency/heading set (Frequency Domain Condition) to be analyzed. This is equivalent to the same reference in a Wadam analysis.

If the HydroModel contains a Morison model, only a time-domain, nonlinear analysis is available. See more information about this in the following chapter.

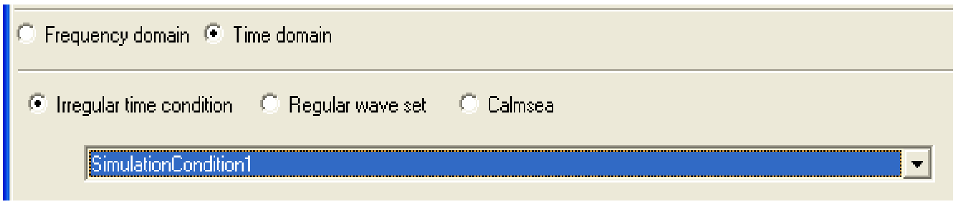


For time domain analysis three different types of input can be specified:

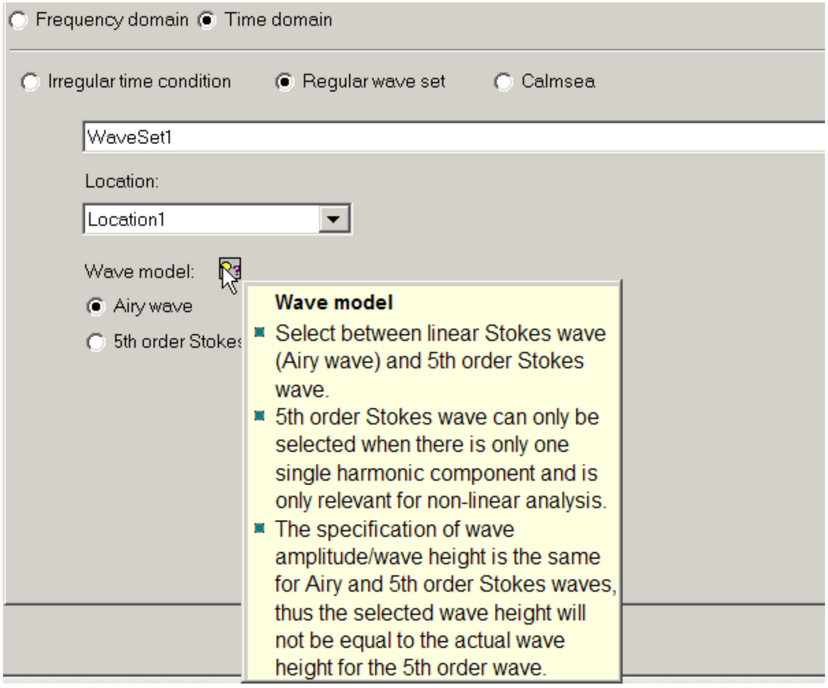
- Calm Sea: No waves given



- Irregular time condition: The wave components are computed by HydroD from a selected Sea state. I minimum wave period should also be given to give a lower cut-off for the wave spectrum.

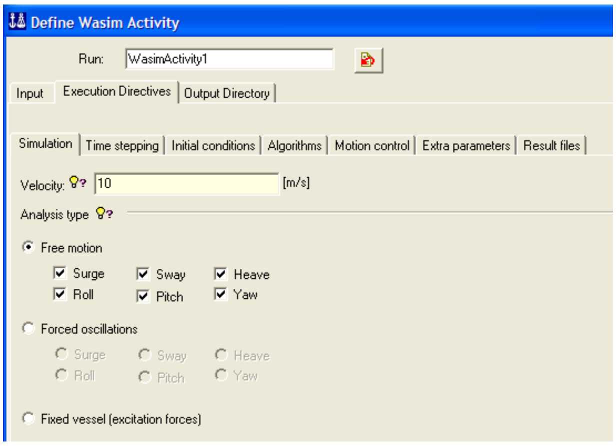


- Regular wave set: The harmonic wave components are defined directly by the user. The user can select between linear Stokes wave model (Airy wave) and 5th order Stokes wave model. 5th order Stokes wave can only be selected when there is only one single harmonic component and is only relevant for non-linear analysis.



The execution directives tab

Simulation

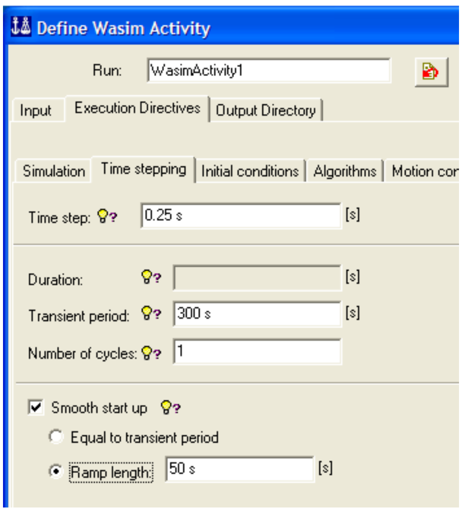


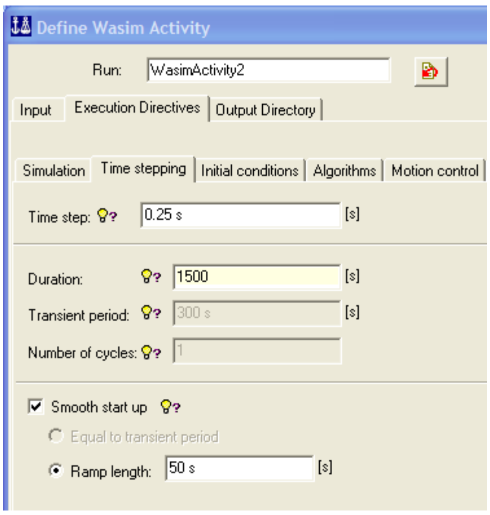
Specify vessel speed and analysis type. The setting here will overwrite the free/fixed setting on the HydroModel.

For the free motion option select which modes that are free to respond. By default all six modes are free.

Forced oscillations can only be done for one mode at the time.

Time stepping





Specify time step and either transient period for frequency domain analysis or duration for time domain analysis. For frequency domain analysis Wasim will compute the simulation length required for the Fourier analysis and add this to the specified transient.

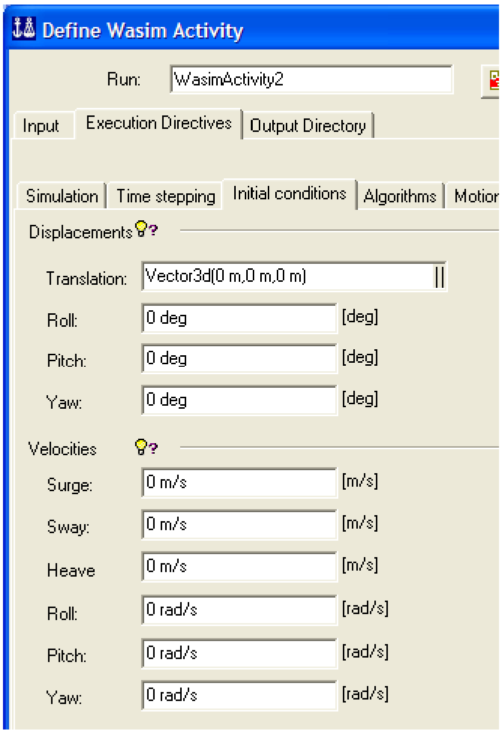
Initial conditions

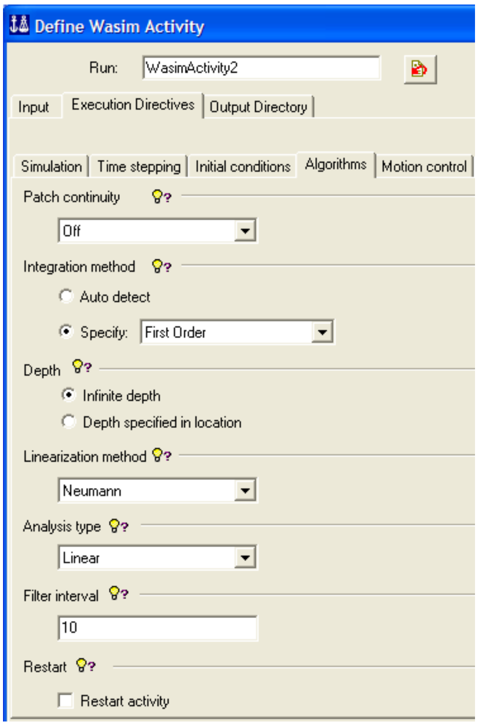
Specify initial displacement and/or initial velocities. This is a useful feature for analysing the motion control system.

Algorithms

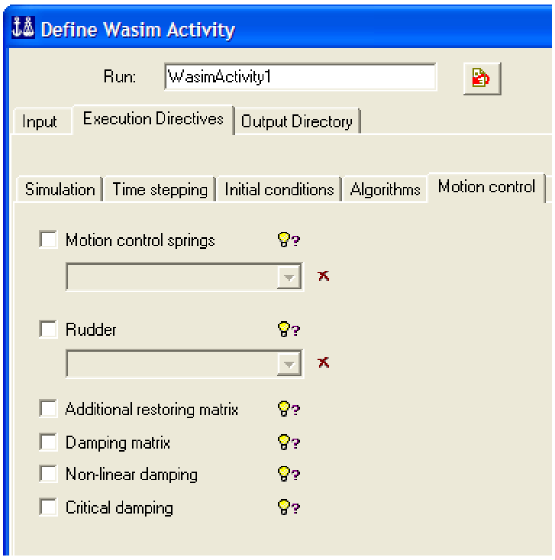
Specify various numerical settings. See light bulbs for details.

If infinite depth is selected, the depth defined in the referenced location will not be used.



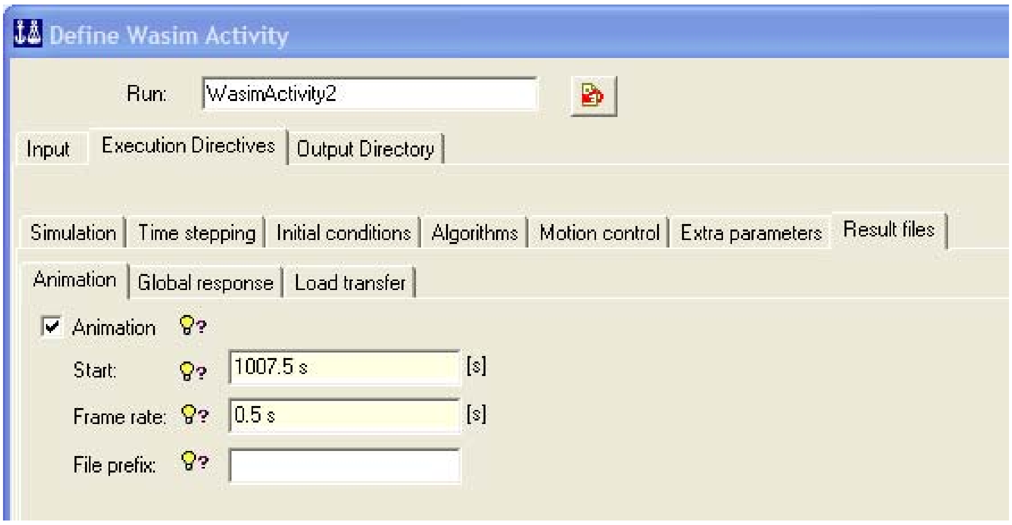


Motion control



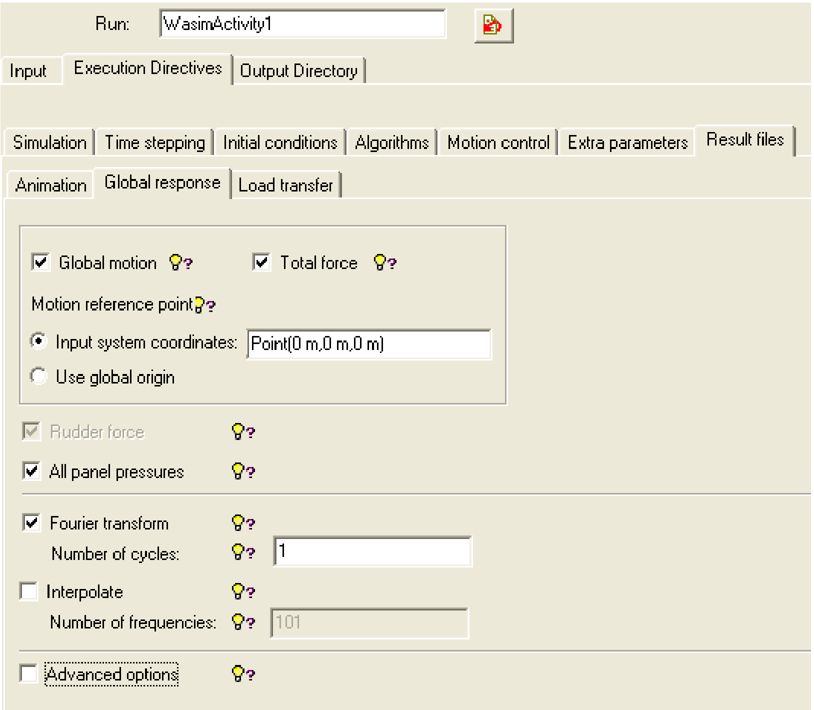
Here you can select which motion control systems you want to include in the analysis. The effect of all selected systems will be added together in the analysis.

Result files



Animation

Define starting point and frame rate for animation output.



Global response

Define output settings for global responses.

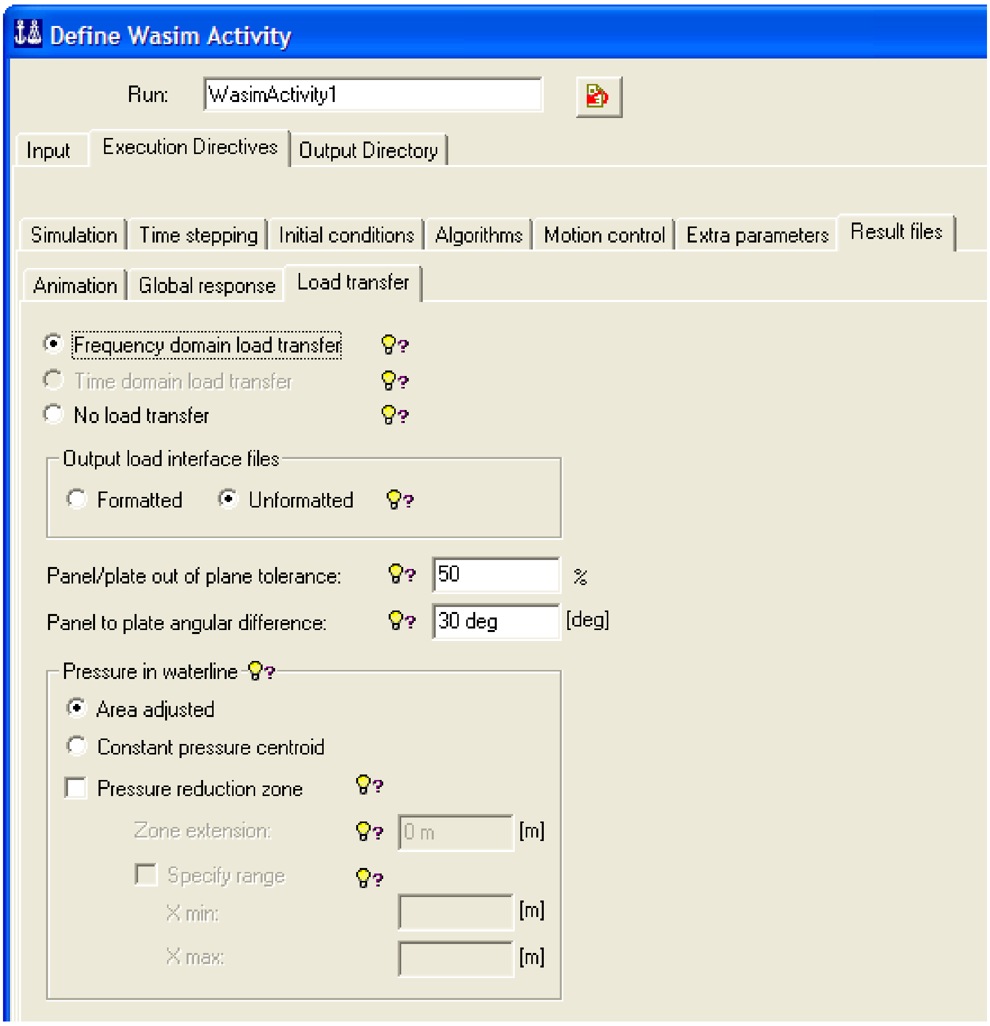
Fourier transform is only relevant for frequency domain analysis.

Remember to tick for **All panel pressures** if load transfer is to be carried out at a later stage.

Motions and forces will refer to the specified **motion reference point**. The motion reference point should be specified in the **input** coordinate system.

The option for interpolation between frequencies should be used with care as this interpolation may introduce significant imbalance in the loads.

Frequency Domain Load transfer



The load transfer in frequency domain from Wasim is identical to the panel to shell load transfer from Wadam (see Wadam Load Transfer).

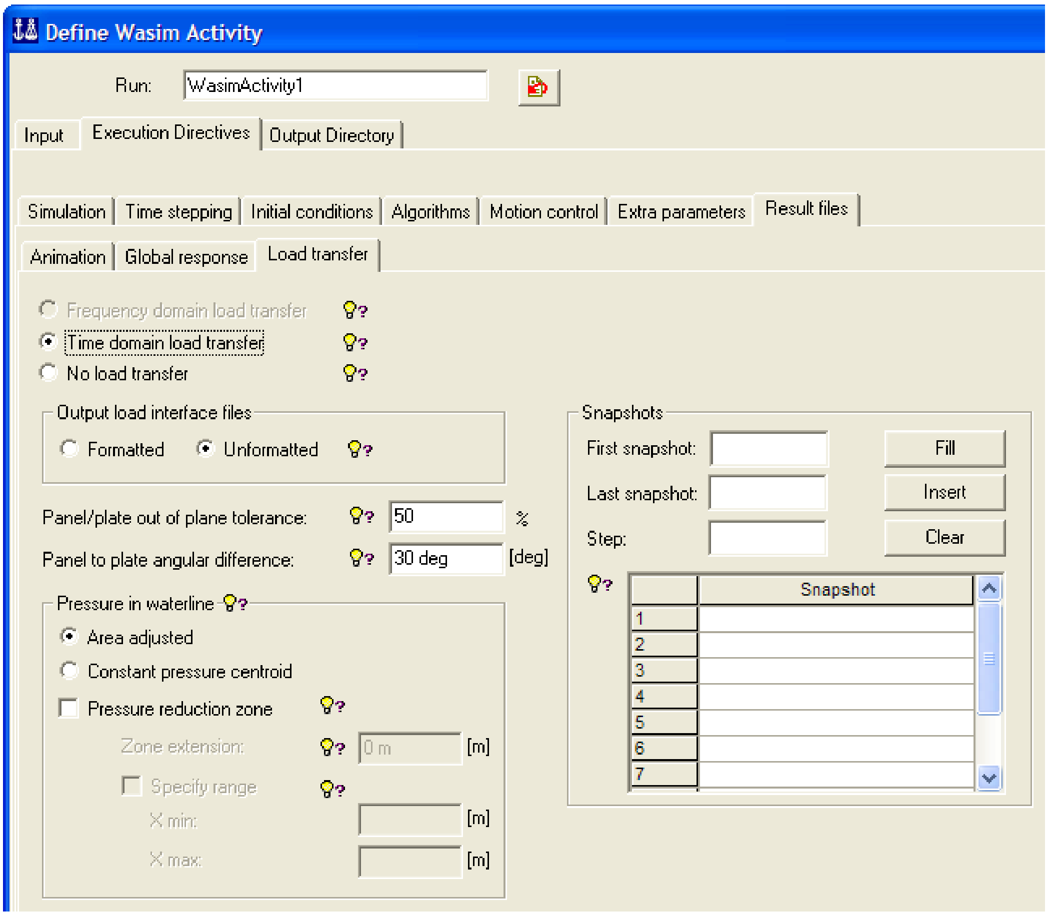
File formats

- Formatted files Load files in text format
- Unformatted files Binary load files (recommended)

Pressure in Waterline

- Constant pressure centroid The panel will receive pressure equal to the pressure at the centroid if the panel centroid is submerged.
- Area adjusted The panel pressure is adjusted according to the submerged area
- Pressure reduction zone The pressure will be modified in a zone around the free surface. Panels above the free surface will get a pressure load, and the pressure on panels below the free surface within the specified zone will be reduced. The pressure reduction zone can be specified for a region on the x-axis. In this case the constant pressure centroid or area adjusted method will be used outside the region.

Time domain load transfer



Specify the points in the time simulation when load snapshots are to be transferred to Sestra. Snapshots loads can be transferred from a linear analysis or a non-linear analysis. The ‘Pressure in waterline’ options are only relevant for the case of a linear analysis.

5.4.3.4 Using Morison models in Wasim

Morison's equation will be included in the Wasim solution when a Morison model is included in the HydroModel. The Morison model can only be used for non-linear, time domain analyses. 2D Morison beam elements defined in the Morison model are supported, as well as anchor and TLP elements. The pressure area elements are also considered to calculate the Froude-Krylov, hydrostatic and buoyancy forces acting on the Morison model.

It is recommended to use HydroD to define the Morison model parameters to be used in the Wasim analysis.

This feature is only relevant to non-linear analysis since the non-linear drag term of the Morison formula is involved.

The Morison model will also be considered for computation of sectional loads and load transfer to the structure model.

5.4.3.5 Handling models in mm in Wasim

The input models (pln file, structure model, etc.) to Wasim can be defined in any of the supported length units. The database length unit should be set when defining a new workspace. It should be noted that all the output from Wasim is given in the database length unit.